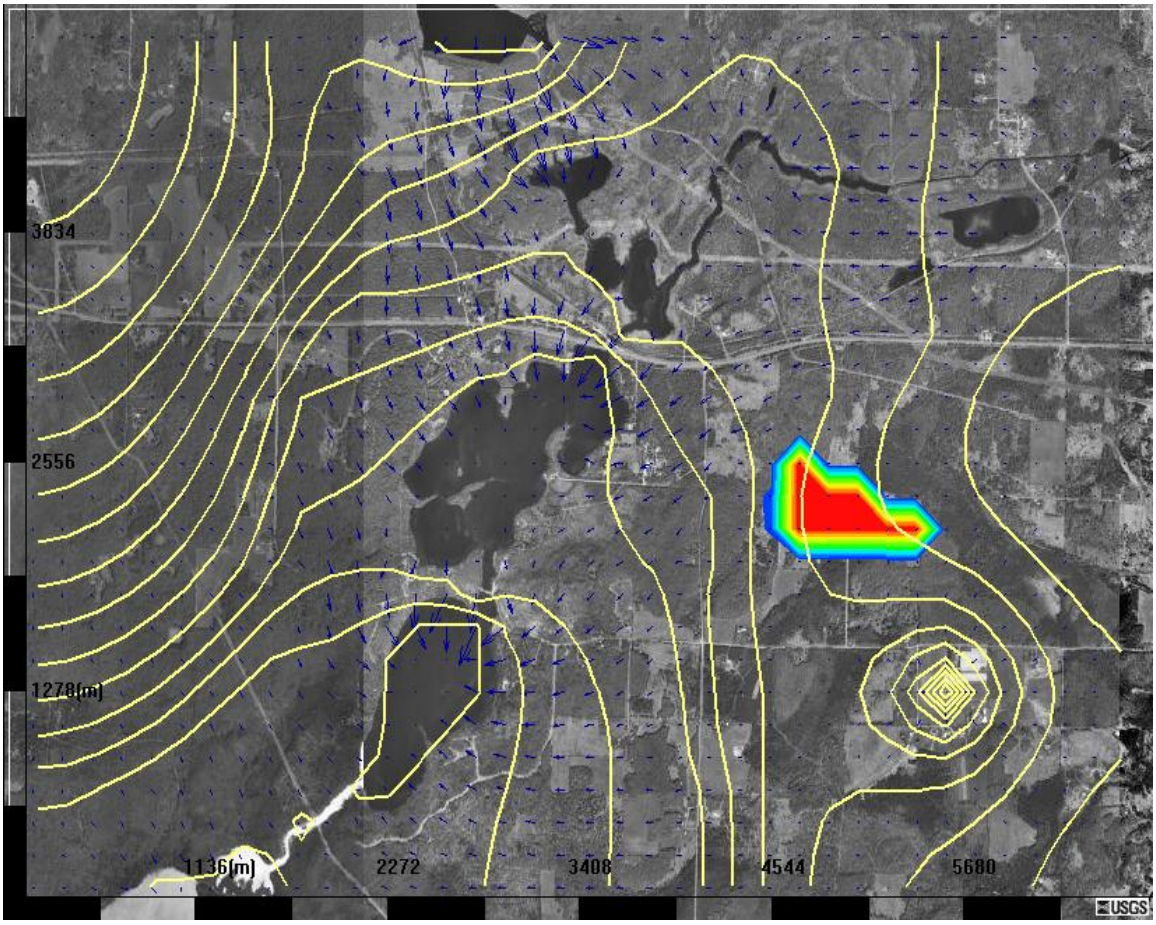
DR. SHUGUANG LI AND ASSOCIATES
INTERACTIVE GROUNDWATER MODELING (IGW)



TUTORIALS – *Version 5.0P*

MICHIGAN STATE UNIVERSITY COLLEGE OF ENGINEERING

Dr. Shuguang Li and Associates at Michigan State University

IGW Tutorials *for Version 5.0P*

Copyright © 2006 by Dr. Shuguang Li and Associates at Michigan State University. All rights reserved.

Dr. Shuguang Li and Associates makes no warranties either express or implied regarding the program IGW and its fitness for any particular purpose or the validity of the information contained in this document. The inputs and outputs described herein do not necessarily represent actual field conditions – they are only used to illustrate the capabilities of IGW.

Authors / Editors:
Hassan Abbas
Dr. Shuguang Li

DOCUMENT VERSION: **2008-1**

http://www.egr.msu.edu/~lishug

TABLE OF CONTENTS

CHAPTER 1: INTRODUCTION 1

1.1. STRUCTURE OF THE TUTORIAL 1
1.2. THE PROBLEM AND THE AVAILABLE DATA 1
1.3. CONCEPTUAL MODEL 2
1.4. COMMENTS 4

CHAPTER 2: STARTING AND EXPLORING IGW 5

2.1. STARTING IGW 5
2.2. IGW INTERFACE 6
2.3. COMMENTS 9

CHAPTER 3: SETTING UP A MODEL 10

3.1. IMPORTING A BASEMAP 10
3.2. DEFINING THE MODEL DOMAIN 13
3.3. ADDING RECHARGE 16
3.4. ADDING A SURFACE WATER FEATURE IN THE MODEL 17
3.5. ADDING WELL FEATURES 19
3.6. SAVING THE CONCEPTUAL MODEL 21
3.7. SAVING THE DISPLAY AS A PICTURE 22
3.8. OPENING A SAVED MODEL 23

CHAPTER 4: OBTAINING A SOLUTION 24

4.1. DISCRETIZING THE MODEL 24
4.2. SOLVING THE MODEL 25

CHAPTER 5: VIEWING MODEL OUTPUTS 26

5.1. VIEWING HEAD AND VELOCITY FIELDS 26
5.2. DEFINING AND VIEWING CROSS-SECTIONS 26

CHAPTER 6: CHANGING DISPLAY OPTIONS 28

6.1. CHANGING DISPLAY IN MODEL AREA 28
6.2. CHANGING DISPLAY FOR CROSS-SECTION 31

CHAPTER 7: EXPLORING THE CURSOR ACTIVATED TABLE 33

7.1. THE CAT INTERFACE 33
7.2. ADDING/REMOVING FIELDS IN CAT 34

CHAPTER 8: EXPLORING THE ATTRIBUTE EXPLORER WINDOW 35

8.1. THE ATTRIBUTE EXPLORER WINDOW 35
8.2. MODEL EXPLORER TAB 35
8.3. ATTRIBUTE INPUT PANE 36
8.4. EDITING FEATURES IN MODEL EXPLORE PANE 38

CHAPTER 9: PARTICLE TRACKING 39

9.1. FORWARD PARTICLE TRACKING 39
9.2. BACKWARD PARTICLE TRACKING 42
9.3. INITIALIZING THE PARTICLES 42
9.4. INITIALIZING THE CLOCKS 42

CHAPTER 10: ADDING COPLEXITY TO THE MODEL 43

CHAPTER 11: ADDING LAYERS 44

11.1. ADDING A NEW MODEL LAYER 44
11.2. EDITING LAYER ATTRIBUTES 46
11.3. EDITING OBJECTS IN A LAYER 46
11.4. ADDING COMPUTATIONAL LAYERS 47
11.5. PUTTING PARTICLES IN THE APPROPRIATE LAYER 49
11.6. MODEL OUTPUTS WITH ADDED COMPLEXITY 49

CHAPTER 12: USING SCATTER POINTS 53

12.1. ADDING SCATTER POINTS 53
12.2. VISUALIZING RESULTS 56

CHAPTER 13: REFINING THE MODEL 58

13.1. ADDING MORE FEATURES IN THE MODEL 58
13.2. ADDING DISPERSIVITY 60
13.3. REFINING GRID SIZE 61

CHAPTER 14: MODELING CONTAMINATION PLUMES 63

14.1. MODELING A CONTINUOUS CONCENTRATION SOURCE 63
14.2. INITIALIZING THE PLUME 65
14.3. INITIALIZING THE CONCENTRATION CLOCK 65

CHAPTER 15: UTILIZING A MONITORING WELL 67

15.1. SETTING UP A MONITORING WELL 67
15.2. OBSERVING THE MONITORING WELL 68

CHAPTER 16: UTILIZING MASS BALANCES 71

16.1. SETTING UP AN AREA FOR MASS BALANCE 71
16.2. OBSERVING THE MASS BALANCE 72

CHAPTER 17: VIEWING THE MODEL IN THREE DIMENSIONS 73

17.1. INTRODUCTION AND BASIC FEATURES 73
17.2. THE CROPPING FUNCTION 74
17.3. FENCE DIAGRAMS 75

CHAPTER 18: SIMULATING TRANSIENT CONDITIONS 77

18.1. CHANGING SIMULATION TIME PARAMETERS AND SOLVING 77

CHAPTER 19: CONCLUSION 83

BUTTON PALETTE: 84
REFERENCE FROM USER MANUAL 92

CHAPTER 1: INTRODUCTION

This tutorial is designed to step you through some of the basic features of the IGW software (Version 4.7).

You will get the most out of this tutorial if you work the sections in order, and complete all of the sections within a chapter at the same sitting. It is recommended that you use the suggested values for each step in the tutorial. [The suggested values and other appropriate comments appear in bracketed blue text. Where not specified, use default values, default units, or both.] Using the suggested values will walk you through an example, providing you with results that are intuitive, graphical and instructional. The example is very similar to a real world problem, however, bear in mind that this example is only for illustrative purposes.

Hyperlinks are extensively used in this document and are shown as blue underlined text.

You should consult the IGW User's Manual for more in-depth information concerning program implementation and functions.

For detailed information concerning technical content of the model (i.e., mathematics and theory), please consult the IGW Version 5.0P Reference Manual.

1.1. STRUCTURE OF THE TUTORIAL

This tutorial is designed to help you first setup a simple groundwater model for a given problem (which is described in the next section), then gradually add complexity to the model and visualize model outputs at every step as you go. Chapters 2 will introduce you to the basic interface of IGW. Chapters 3 to 5 will let you develop a simpler conceptual model for the problem at hand, setup a complete working model in IGW and visualize model simulation outputs. Chapters 6 to 8 will let you learn and experiment with more advanced features in IGW interface. Chapter 9 will introduce particle tracking. Chapters 10 to 14 will let you add complexity to the model by adding more layers, refining aquifer dimensions, introducing heterogeneity and modeling/observing contaminant concentrations. Chapter 15 will introduce you to mass balance. Chapter 16 will run you through advanced 3D visualization features of IGW. Chapter 17 will let you learn transient simulation. Chapter 19 will conclude the tutorial. Finally, there are two annexure at the end. The first one is a ready reference for all the buttons on IGW interface. The second one contains a few important references from the IGW User's Manual.

1.2. THE PROBLEM AND THE AVAILABLE DATA

A small community pumps groundwater from a confined sandstone aquifer for their drinking water supply. A monitoring study showed that the aquifer is contaminated by TCE plumes near the Boeing and Cascade sites up-gradient from the pumping wells. An aerial photograph of the area is shown in Figure 1.1.1

Although the well water is currently not contaminated, the citizens are extremely concerned since groundwater is their exclusive source of water supply. The fact that groundwater is invisible makes them particularly anxious and they have been asking many questions:

1. Will the advancing plume hit the community wells? How long does it take for the plume to reach the wells?
2. Can the community still use the groundwater? Will the community pumping affect the plume migration? What is the influence area of the community wells?
3. What is the area of contribution (recharge area) of the community wells?
4. How can the community develop a wellhead protection program that protects the community water supply?

A groundwater model can be used to answer such questions. In this tutorial, we shall systematically build a groundwater model to answer the questions raised by the community.

To build any model, data on aquifer properties, aquifer dimensions, sources and sinks, and concentration/location of contaminants is required. Data from site investigations and other sources is available for this site.

Section AA′, as marked on Figure 1.1.1, is shown in Figure 1.2.1. We shall use this and other information to build the model.

***Note*: The model parameters, data and site characteristics used in this tutorial problem are for illustrative purposes and do not represent reality.**

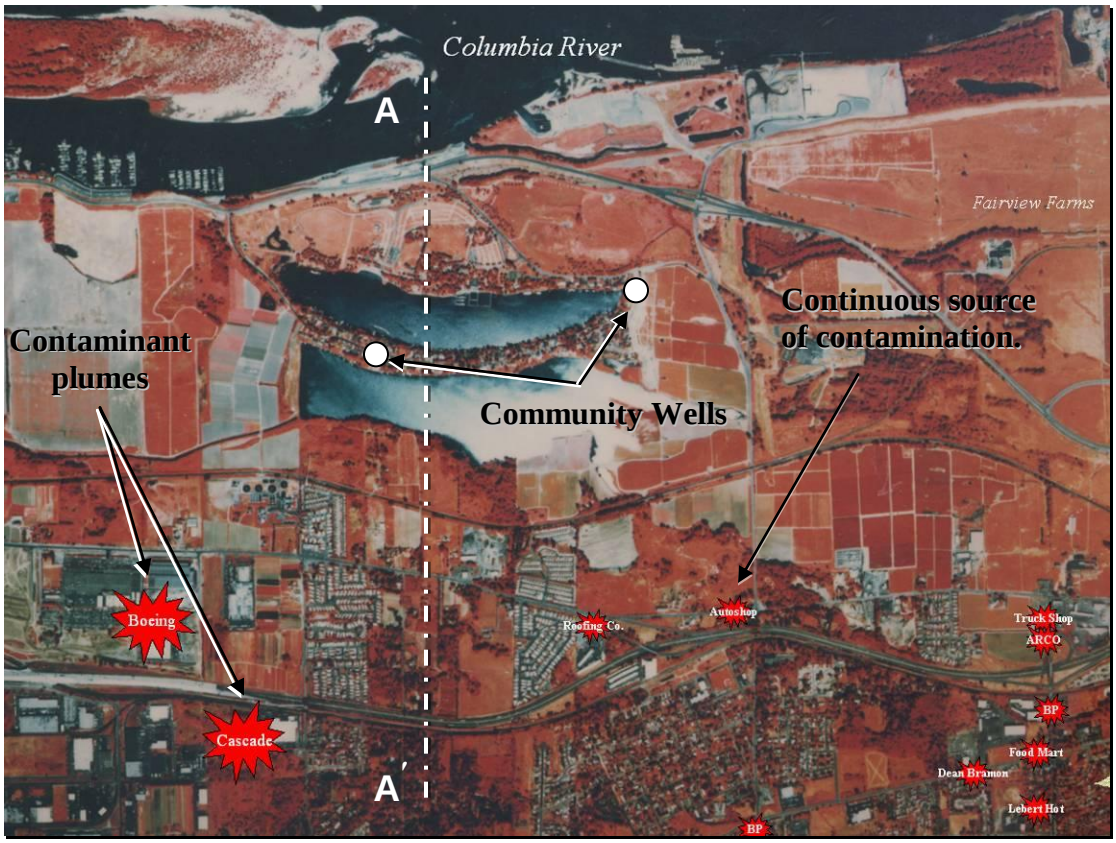


FIGURE 1.1.1: Map of area showing pumping well locations and contaminant plumes

1.3. CONCEPTUAL MODEL

Based on the available information, we build our conceptual model. The conceptual model includes aquifer components (geological layers), sources and sinks and the boundary conditions.

The site has three distinct geological layers. The top layer is low conductivity overburden, approximately 20m thick. This is underlain by high conductivity sandstone layer, approximately 40m thick. Finally, there is a very low conductivity layer of undifferentiated sediments under the sandstone. This layer, as shown in figure 1.1.2, pinches outward to the surface near A′.

Other significant components of our model are Columbia River, Blue Lake, Fairview Lake and pumping wells. Conceptualization of these components along with geological layers is shown in Figure 1.2.1.

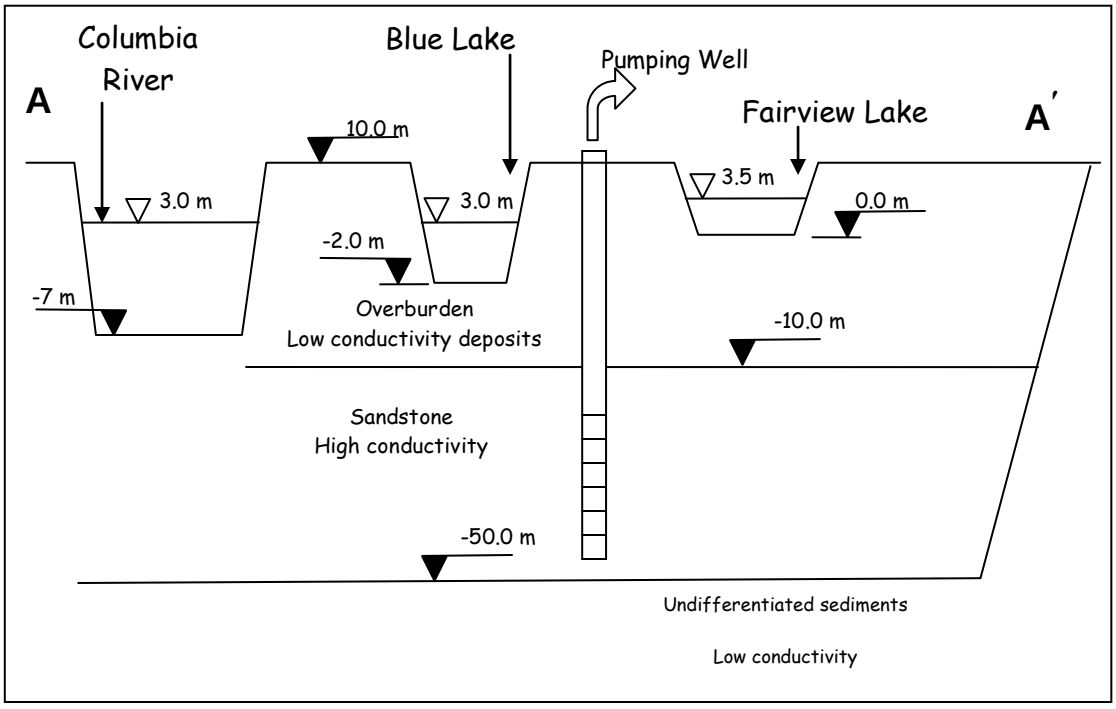


FIGURE 1.2.1: Conceptual layers and aquifer components.

Aquifer properties and conceptualization of sources and sinks in the system are illustrated in Figure 1.2.2. Sources in the system include annual deep percolation (recharge) and leakance from lakes. Columbia River and pumping wells are identified as the sinks in the system. However, Columbia River can act as source if head in the aquifer drops below the stage of the river

Contamination concentrations are visualized as plumes in the sandstone layer – which represents the main aquifer in our conceptual model.

Boundary conditions for the north and south of the model (corresponding to A and A′ respectively) are also shown in Figure 1.2.2. Boundary to the north is taken as constant head boundary (a model boundary at which the head does not change), equal to the stage of the Columbia River. This approximation is based on the fact that Columbia River is a large water body and its level will not be affected by sources and sinks in the model area. The southern boundary is assumed to be a no-flow boundary (a boundary across which no flow can take place). This approximation is based on the assumption that since the layer of undifferentiated sediments (very low conductivity) pinches out to the surface, it effectively blocks any flux entering the aquifer from the southern boundary. The east and west boundaries of the model are also assumed as no-flow boundaries. Although the aquifer continues without a geological feature physically separating the model area in either eastern or western directions, the direction of the regional flow is approximately parallel to these boundaries. If we assume there is no lateral flow across these boundaries, these can be assumed, in the model, as no flow boundaries. However, while using our model, we should bear in mind these are ‘soft’ no-flow boundaries. Our no-flow assumption will not hold true if there are large stresses close to these boundaries – resulting in increased prediction uncertainty.

Figure 1.2.2 below shows the modeling layers, boundary conditions (north and south only), sources and sinks, and aquifer properties.

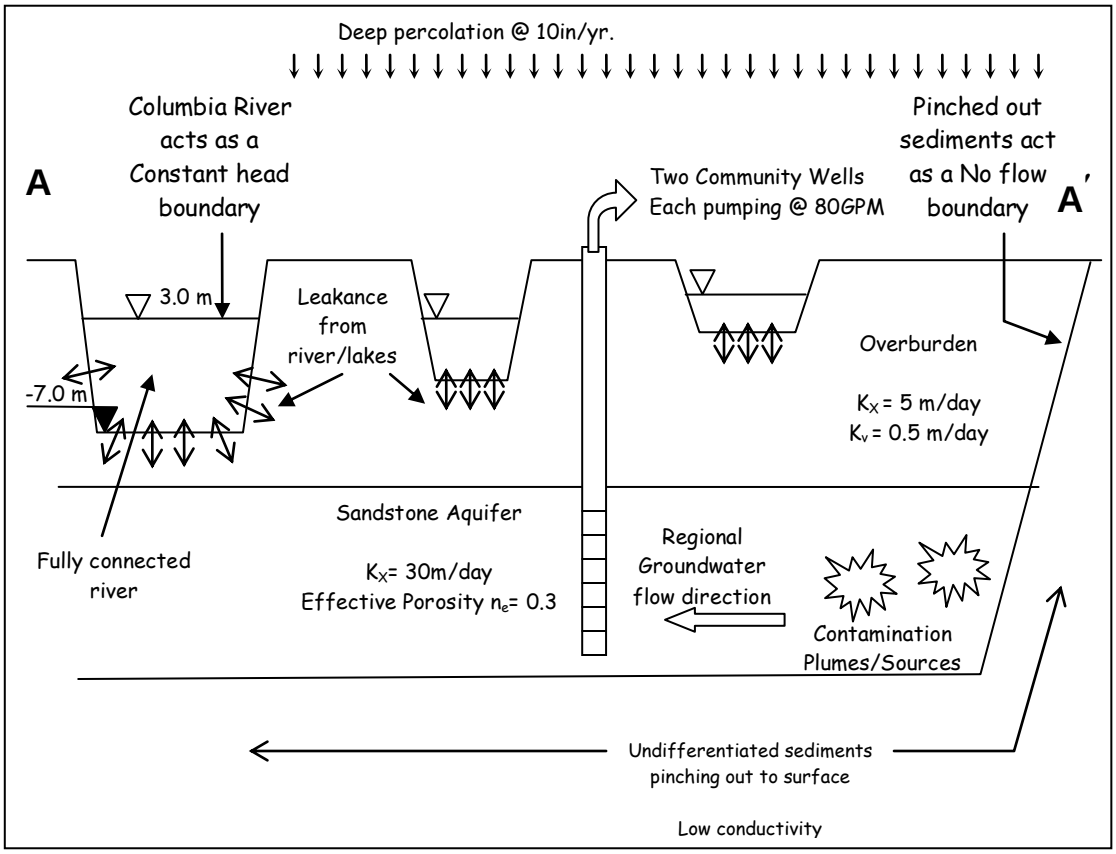


FIGURE 1.2.2: Conceptual components of model.

1.4. COMMENTS

Now that you have a conceptual model of the system, and a list of questions to be answered, you can build the numerical model using IGW 5.0P and use the model simulations to answer the questions. This tutorial will help you learn how to build the model and how to visualize the results.

In the next chapter, you will start the tutorial by familiarizing yourself with IGW main window and working menus. Following chapters will guide you through a step by step tutorial that will let you build the model and simulate results. You will start with a simple model and add complexity to it as you go. The tutorial will give you the flexibility of experimenting and exploring the capabilities of IGW 5.0P.

CHAPTER 2: STARTING AND EXPLORING IGW

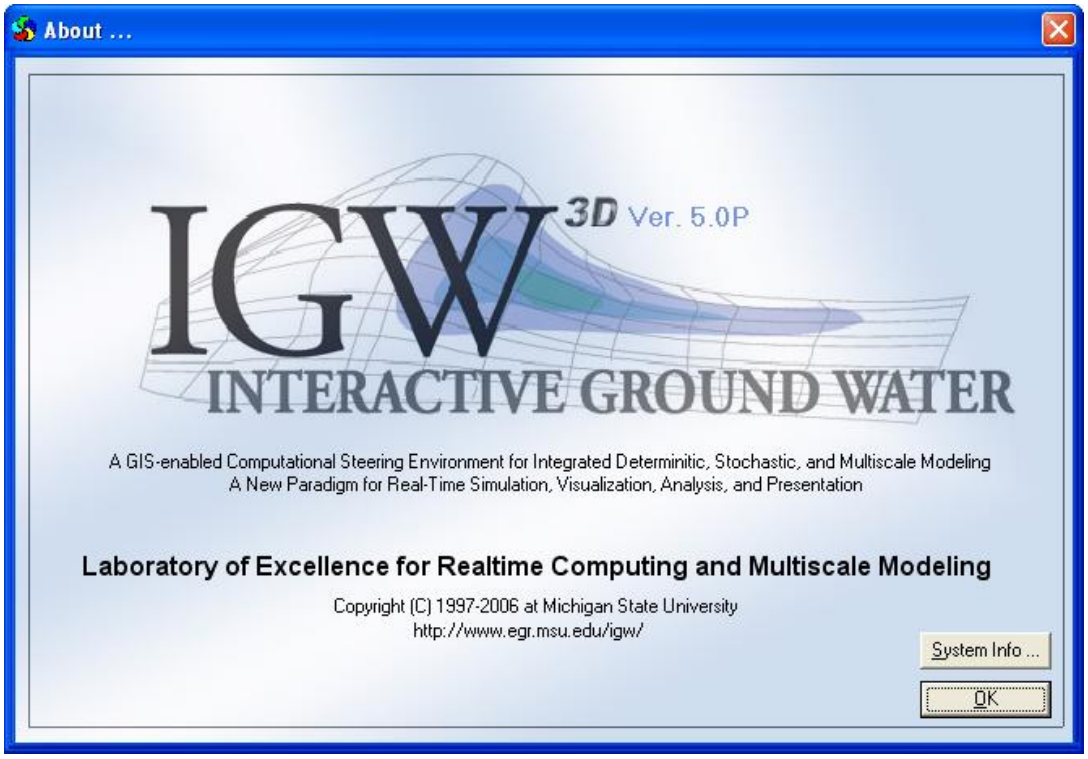
This chapter will show you how to start the program, and give you an overview of the software interface. If you are using the interactive hypertext version of this tutorial then you have already started the software and you may continue to **Section 2.2**.

2.1. STARTING IGW

The easiest way to start the program is to use the Windows™ Start menu.

Step 1: Click on the Start button, open the Programs sub-menu, then the Interactive Groundwater sub-menu, and then select IGW 5.0P (This assumes the software has been installed correctly and using the default paths.)

You will see a splash screen with the software credits. You can either wait 15 seconds for the screen to expire, or click on the 'OK' button to continue.


About ...
IGW 3D Ver. 5.0P
INTERACTIVE GROUND WATER
A GIS-enabled Computational Steering Environment for Integrated Determinitic, Stochastic, and Multiscale Modeling
A New Paradigm for Real-Time Simulation, Visualization, Analysis, and Presentation
Laboratory of Excellence for Realtime Computing and Multiscale Modeling
Copyright (C) 1997-2006 at Michigan State University
http://www.egr.msu.edu/igw/
System Info ...
OK

A full-screen IGW window appears. See Figure 2.1.1

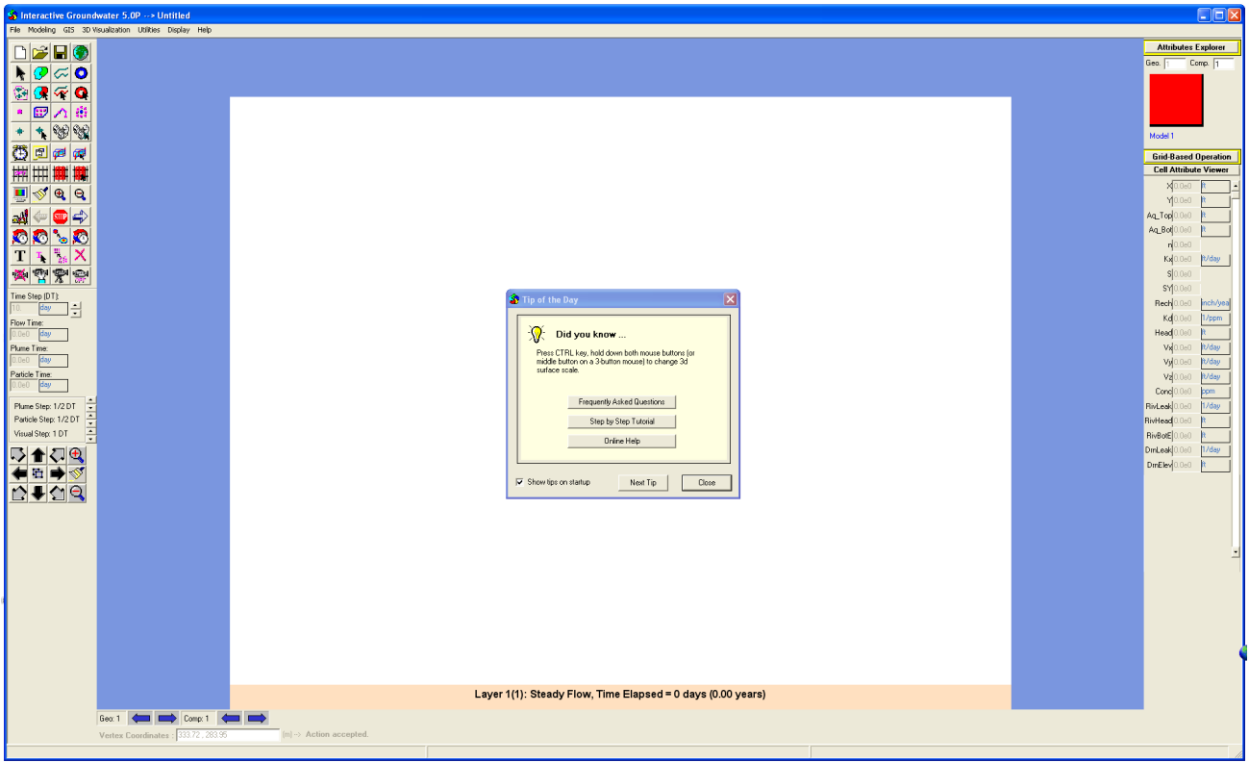

Interactive Groundwater 5.0P -- Untitled
Tip of the Day
Did you know ...
Frequently Asked Questions
Step by Step Tutorial
Online Help
Show tips on startup
Next Tip
Close
Layer 1(1): Steady Flow, Time Elapsed = 0 days (0.00 years)

FIGURE 2.1.1: Start up window for IGW 5.0P

2.2. IGW INTERFACE

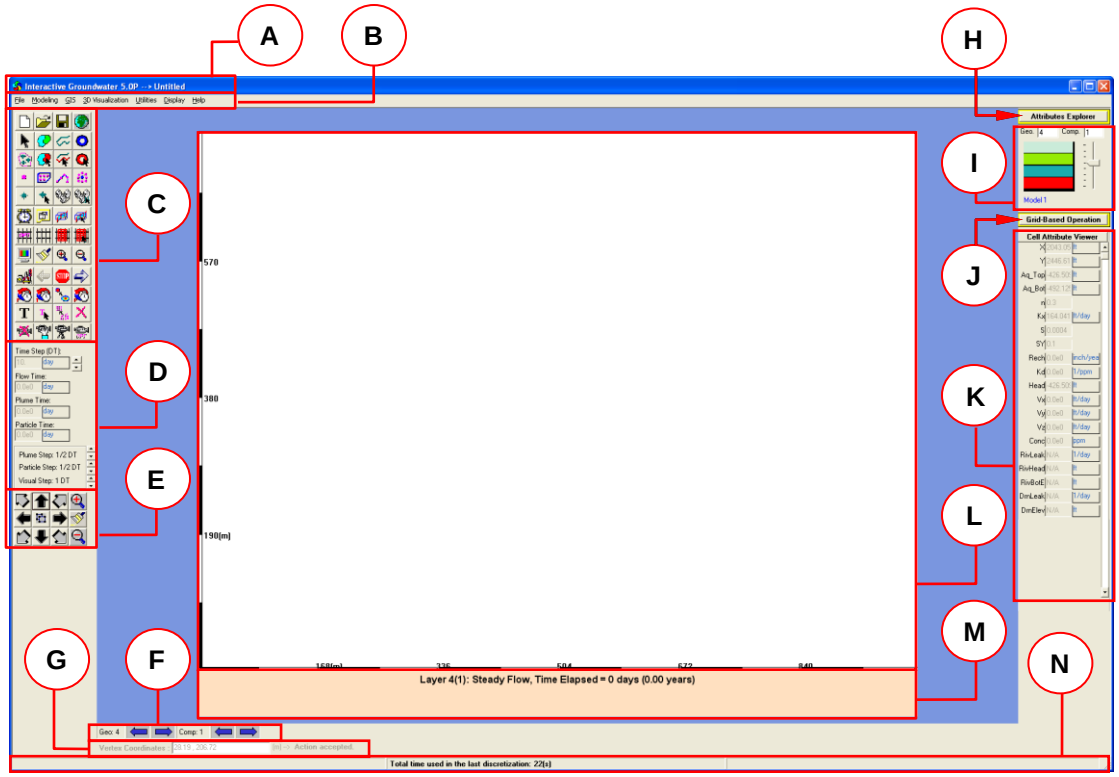
The first thing you notice is the 'Tip of the Day' Window, which gives you a new tip every time you start the software. If you want to disable this function, deselect the 'Show tips as startup' box before closing the window. You can also access the Frequently Asked Questions file, access the hypertext version of this tutorial, or open the Help file by clicking the appropriate button.

You may also access the Help file at any time by selecting 'About…' from the 'Help' menu on the main screen menu bar.

Step 1: Click the 'Close' button when you are finished with the 'Tip of the day' window.

The entire IGW interface is now completely visible. Various portions of the screen are highlighted and annotated in Figure 2.2.1.

The middle of the IGW window is occupied by the 'Model Screen'. The white rectangle within the Model Screen is referred to as the 'Working Area'. This is where you define features, assign attributes, and obtain solutions.



A- Title Bar
B- Menu Bar
C- Button Palette
D- Step Adjustment and Time Display Interface (SATDI)
E- Working Area Display Tools
F- Layer Navigator
G- Vertex Coordinates Interface
H- Attributes Explorer Button
I- Layer Selector
J- Grid-Based Operations Button
K- Cursor Activated Table (CAT)
L- Working Area
M- Working Area Attributes Display (WAAD)
N- Status Bar

FIGURE 2.2.1: Main window for IGW 5.0P

The peach colored rectangle below the Working Area is referred to as the 'Working Area Attribute Display' (WAAD). The WAAD displays information concerning the flow solution and the elapsed time (being the maximum value of the flow time, particle time, and plume time) in the solution for the Working Area within the Model Screen. You may click in the WAAD and subsequently type in information to augment the software title for the present model state.

There is a small box below the Model Screen indicated by the phrase 'Vertex Coordinates Interface' (VCI). This is known as the Coordinate Locator and allows the user to manually type in coordinates used to define a feature in the model (as opposed to clicking the mouse in the Working Area).

Most of the left-hand portion of the window contains a Button Palette. This palette provides you with one-click access to the most commonly used features in IGW. Where ever you see one of the button icons in this document, [*ctrl + click*] will take you to its explanation/definition. Windows standard 'hovering' feature is available for all buttons on the palette. A bubble-box briefly appears near the cursor when the cursor is placed over a button. This bubble-box is informing you as to the functionality of the button.

Title Bar and Menu Bar explained in User's Manual Section 3.2. Layer Selector, Step Adjustment and Time Display Interface (SATDI), Working Area Display Tools and Cursor Activated Table (CAT) are explained in this document as and when these are encountered during the tutorial.

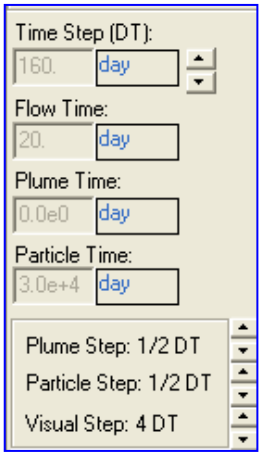
Step 2: Clicking on 'create a new project' button (row 1, column 1) button will open a fresh working area window to start a new project

By default, IGW opens with a new project window and you may not have to use this button when you open IGW.

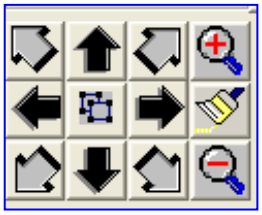
.

Step 3: Clicking on the 'Reset toolbar buttons state' button at any time will reset the cursor and allow you to select other buttons

Below the button palette is the 'Step Adjustment and Time Display Interface' (SATDI). The SATDI provides you with quick access to computational and display adjustments. The time step (DT) (10 by default) can be adjusted using the up/down buttons next to the units list field (day by default). The plume step, particle step, and visual step can be set in a similar fashion, as a ratio of the DT. The Flow Time, Plume Time, and Particle Time sections display the computational time for the flow, plume, and particles, respectively.

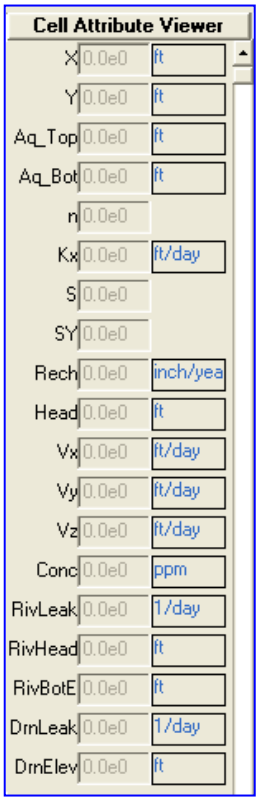


The 'Working area display tools' (WDT) are located below the SATDI. The WDT allows the user to adjust the size and location of the Working Area (and associated WAAD) within the Model Screen. This feature is especially useful for viewing solutions at a higher resolution.

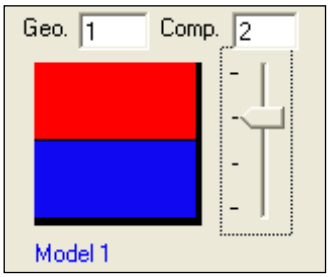


Step 4: Practice using the WDT. Adjust the Working Area (and WAAD) so that it is centered in the Model Screen, and occupies the highest percentage possible of the Model Screen.

The major portion of the right-hand side of the window is occupied by what is known as the 'Cursor Activated Table' (CAT), shown in the graphic on the right. This table will display the variable values that exist in the model at any point the cursor is located. It is more intuitive to discuss the CAT with a working model, so that is done in **Chapter 7**.



The 'Layer Selector' is located above the CAT. It provides the user with quick access to different geological and computational layers that are defined in a model. The Layer Selector is discussed more in **Sections 11.5 and 11.6**.



2.3. COMMENTS

This chapter was designed to give you a brief overview of the IGW Version 4.7 interface and help options. You now have the basic skills to complete the rest of the tutorial sections.

While we have not explored the functionality in any depth yet, one of the powerful features of IGW is that it is very intuitive. This quality helps you grasp the functionality, by its design incorporating a logical layout of both the software and its interface. As a result, we will rely on the program's user-friendly design to help you understand the functionality better.

The following chapters will employ that intuitive quality in a series of directed steps that will culminate in a solution to a typical groundwater problem. When finished, you will have solved a complex groundwater contamination problem with software you have only just begun to discover!

CHAPTER 3: SETTING UP A MODEL

Our aim in this tutorial is to begin with a very simple model and gradually add complexity as we explore IGW. This chapter will walk you through the basic procedures for setting up a simple model in IGW 4.7.We will develop a simpler model by making the following simplifying assumptions in the conceptual models shown in Figures 1.2.1 and 1.2.2.

- The two geological layers can be lumped into a single layer.
- Being smaller water bodies, the impact of Blue Lake and Fairview Lake on the aquifer can be ignored.
- The only features of hydrological significance are Columbia River and pumping wells.

The simplified conceptual model is shown in Figure 3.0.0

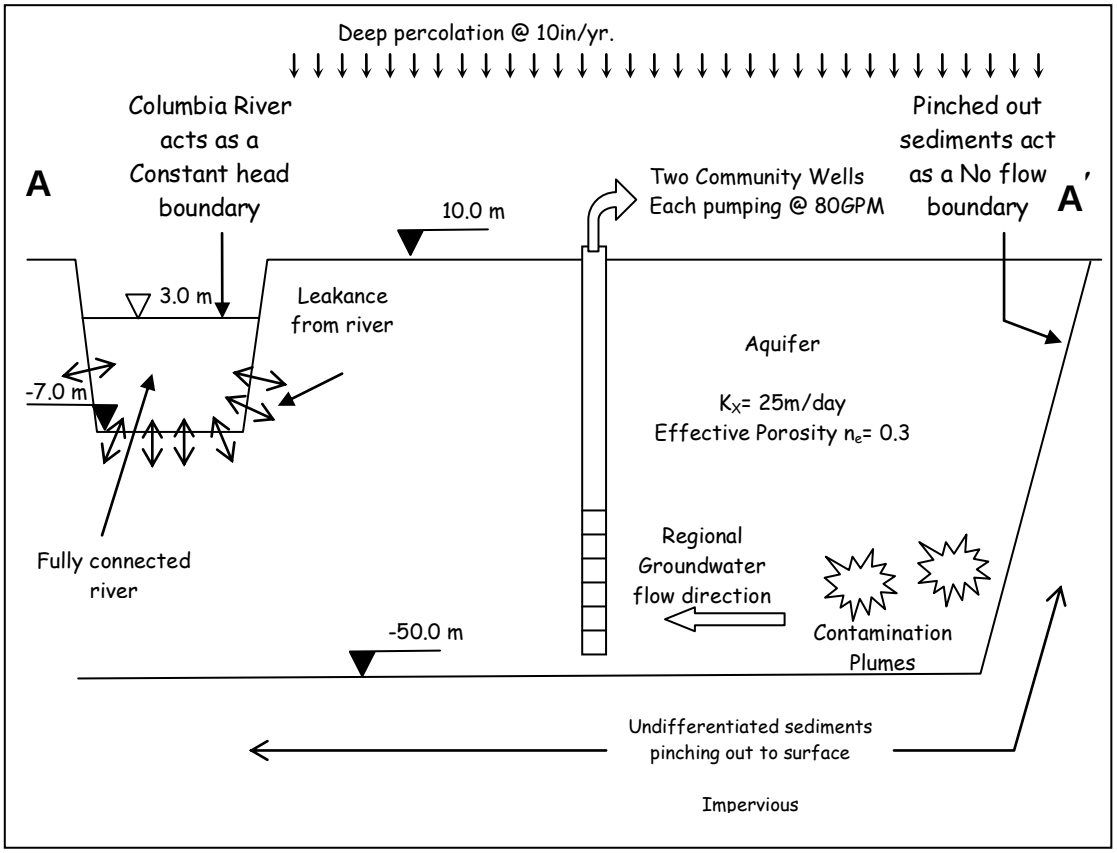


Figure 3.0.0: Simplified Conceptual Model

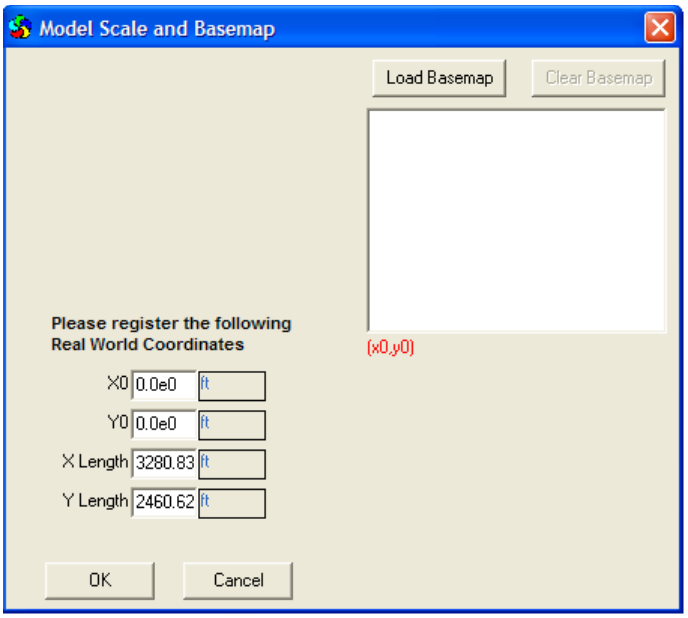
3.1. IMPORTING A BASEMAP

The first step in defining any model is to know its associated real-world characteristics. Let's begin with the plan-view location.

Most of the procedures in defining model features will be done in the Working Area, so let us first import the aerial map in Figure 1.1.1 in the Working Area to use as a basemap in defining our model features.

Step 1: Select the 'Set Domain and Register a Basemap' button, located at Row 1, Column 4 of the button palette.

This brings up the 'Model Scale and Basemap' window.



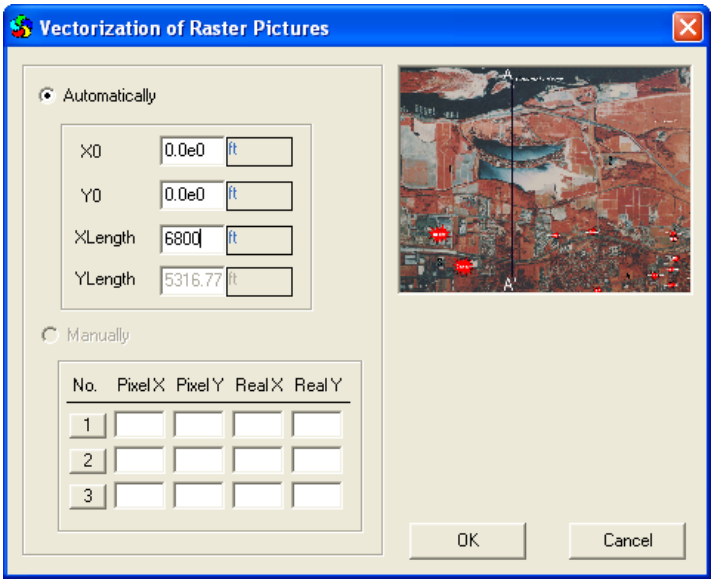
Step 2: Click the 'Load Basemap' button. This brings up the 'Open' window, in which you can navigate to any files within your computer.

Step 3: Browse to the location that your basemap is located, select it, and click the 'Open' button.

The suggested basemap is located at:
http://www.egr.msu.edu/hydrology/Research/software igw trial version.htm

If you open a BMP, GIF, or JPEG file, the 'Vectorization of Raster Pictures' window appears (go to Step 3). If you open a SHP or DXF file, go to Step 7 (these files do not need to be vectorized).

Step 4: Set the origin coordinates and the image lengths corresponding to the real world. At a minimum, you should set the XLength [6800 ft] as the horizontal distance that the basemap covers in the real world.

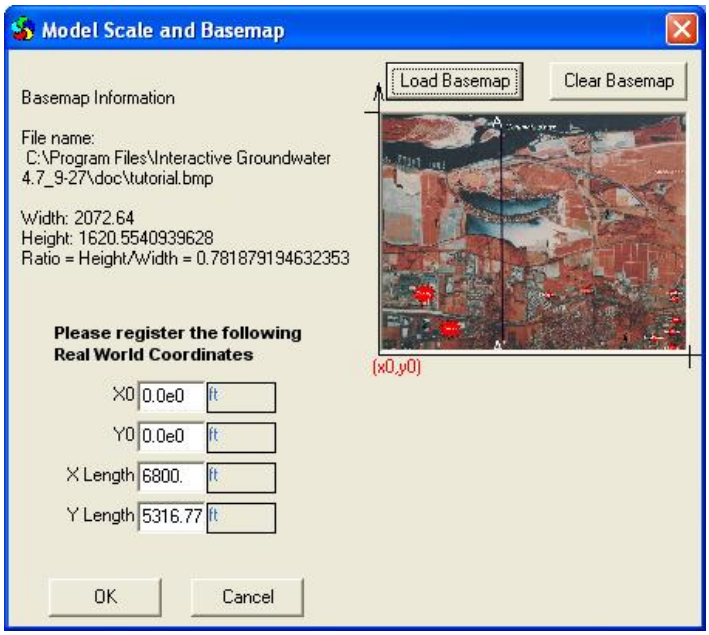


(*Note*: First change the units and then input the numerical value. When you change units, IGW automatically converts the existing numerical value into the new units, as seen above when 6800 ft is entered as the x-value. Getting in this habit will prevent input errors when inputting data into IGW.)

Also, after changing units, be sure to delete all numbers from the numerical field. Some numbers may be present in the field, but not visible, due to the number having many decimal places. Again, this helps prevent data errors. The Y-Length will be calculated automatically, based upon the dimensions of the image. X0 and Y0 are coordinates for the left, bottom most corner of the picture. These are set to zero as the default [use the default values].

Step 5: Click 'OK'.

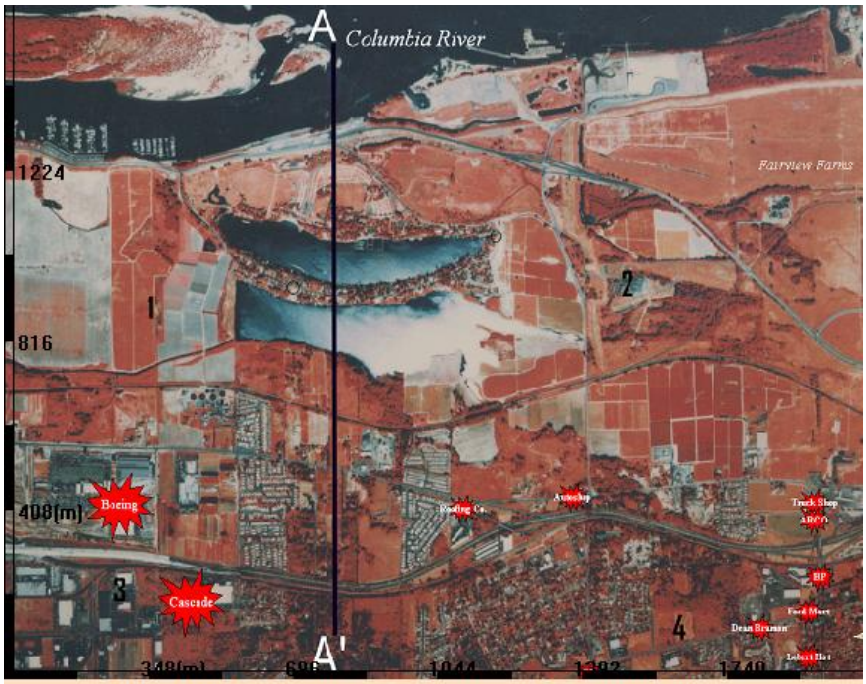
Step 6: The 'Model Scale and Basemap' window becomes visible again with the selected basemap as a preview. Here you can also make adjustments to the scale and coordinates for the files.



(*Note*: If you change the Y Length value the picture does not distort, but the software does display extra space as white in the Working Area.)

Step 7: Click 'OK'.

The program returns to the main screen with the desired basemap as the background in the Working Area.



IGW also allows you to bring in multiple files. Simply repeat the steps above to bring in another file, and it automatically merges with the present basemap. This means that the pictures are both visible and scaled but now become one image in the software. Therefore, if you import an incorrect picture or assign the wrong scale, you will have to clear the entire merged basemap set. Once the basemap is imported you can define areas and features that correspond to it.

3.2. DEFINING THE MODEL DOMAIN

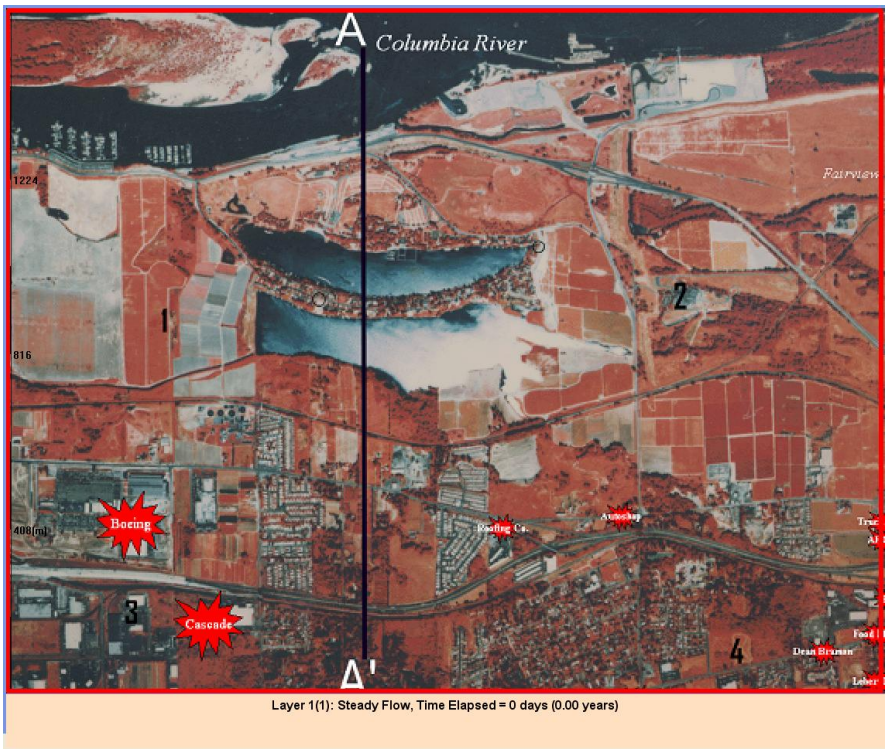
The first step in modeling with IGW is to define a 'parent zone' that encompasses the area over which the solution will apply. We generally would like to define this zone with boundaries at places where water does not flow through, or where the head is known. In hydrogeology, these are known as the **no-flow boundary conditions** or **constant head boundary conditions** respectively. We already discussed boundary conditions in Section 1.2. (Please refer to Section 7.6 of User's Manual for more details on assigning zone attributes).

The problem area is now represented by a base map in IGW. Once you have decided about the model boundaries you can define the parent zone along these boundaries. Having decided the model boundaries, you can proceed to Step 1.

Step 1: Click the 'Create New Arbitrary Zone and Assign Property' button, located at row 2, column 2 of the button palette.

The cursor is now initialized to add zones in the working area.

Step 2: Create a zone that corresponds to your pre-determined boundaries [in this case choose the entire working area]. A zone is created by selecting points cyclically around the area you wish to enclose as the zone. Click the mouse at a location on the zone boundary. Move to another location and repeat. When you have finished, double-click the mouse.



(*Note*: You can easily define a rectangle by holding down the shift key before clicking the mouse on the first point and then simply moving the cursor to the opposite end of the rectangle and clicking the mouse again.)

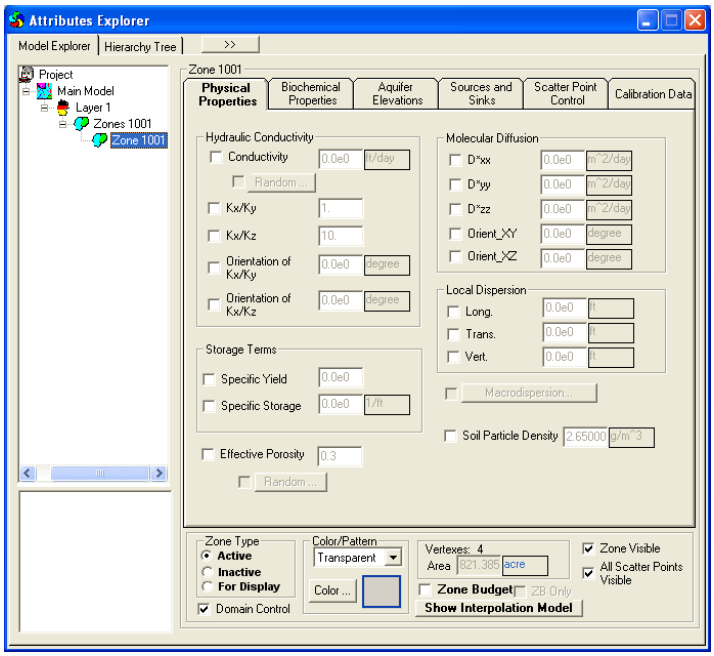
A zone is now defined that covers the entire area of the model screen. This zone will be referred to as the 'parent zone'. The edges of the zone will now appear red in the model screen, indicating that the zone is selected (any modeling feature when it is 'marked' or 'selected' in IGW Working Area appears in bold face red).

Step 3: Assign attributes for the newly defined zone. Click the 'Attribute Explorer' button in the top right corner of

IGW screen to access the Attributes Explorer Window (AE). Alternately, drag the window from the lower right-hand corner of the Windows™ screen and into the middle.

AE is explained in greater detail in **Chapter 8** . Even more details are given in IGW User's Manual.

In the Attributes Explorer window, you can see that 'Zone 1001' is highlighted in the left-hand pane. This corresponds to the edges of the zone highlighted with thick red line in the working area (i.e. it has been selected). Zone 1001 is the default name assigned to the zone. You can edit the name by clicking once on it and then typing any name you wish. For this tutorial, we will go with the default name. Since 'Zone 1001' is the parent zone for the model, you should assign it as 'Domain Control' zone by checking in the box at the bottom left of the AE window as shown below.



Step 4: Perform the following steps to assign conductivity and elevation values:

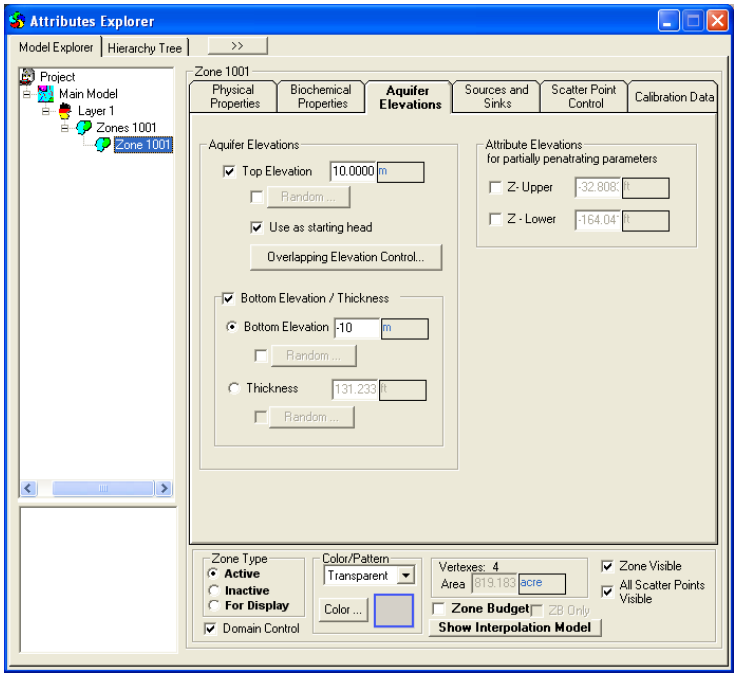
a) Notice the six (6) tabs for Zone 1001, allowing the user to define various properties of the zone.

b) Under the "Physical Properties" tab, check the box next to 'Conductivity'. Before entering a value for conductivity, make sure you have the right units selected. To select the units, just keep clicking on the blue unit box until the desired units [m/day] comes up. Enter the value [20] in the required field **after** the units are set.

c) Click the 'Aquifer Elevations' tab.

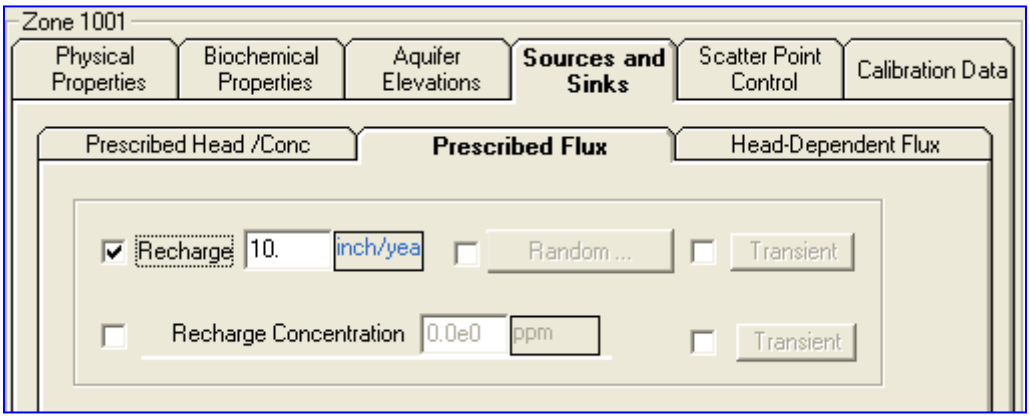
d) Check the 'Top Elevation' box, change units to [m], and enter a value of [10].

e) Check the 'Bot Elev / Thickness' box. Check the radio button for 'Bottom Elevation' (or make sure it is checked), change unit to [m] and enter a value [-50] in the associated field.



3.3. ADDING RECHARGE

Recharge can be added to the model in two different ways. First, recharge can be modeled on a zone basis by accessing the 'Sources and Sinks' tab of Zone 1001 in the AE. There are three (3) tabs associated with this subsection (Prescribed Head, Prescribed Flux, and Head-Dependent Flux). Recharge value are entered under the 'Prescribed Flux' tab, in the 'Recharge' field.



This method may present problems if you have more layers in the model (that you will later add as you go with the tutorial) when trying to model precipitation recharge, due to the possibility of layers becoming dry (and hence adversely affecting the model solution - a detailed description of this problem is not warranted in this tutorial though found in the Users Manual).

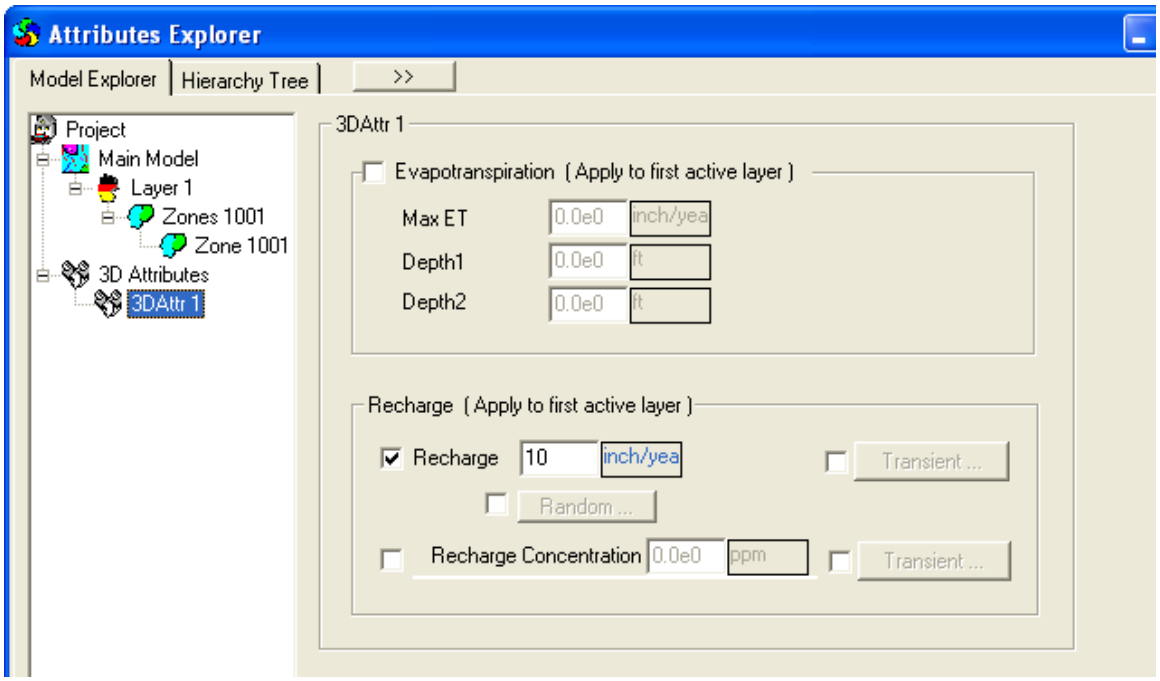
A more useful method for modeling precipitation recharge is to apply recharge to the first *active* layer. This is presented in the following steps. (*Note*: this process is not layer-specific and therefore does not depend on which layer is active while performing these steps.)

Step 1: Select the 'Add 3D Attribute Model' button. The cursor is now initialized to add a 3D-attribute feature.

Step 2: Define a feature that encompasses the entire Working Area (same extent as the parent zone).

Step 3: Access this new feature in the AE, under '3D Attributes'.

Step 4: In the 'Recharge (Apply to first active layer)' section, click on 'Rech', adjust the units [inch/year], and enter a value [10].



The software will now apply this recharge to the first active layer, over the extent of the 3D-attribute feature.

3.4. ADDING A SURFACE WATER FEATURE IN THE MODEL

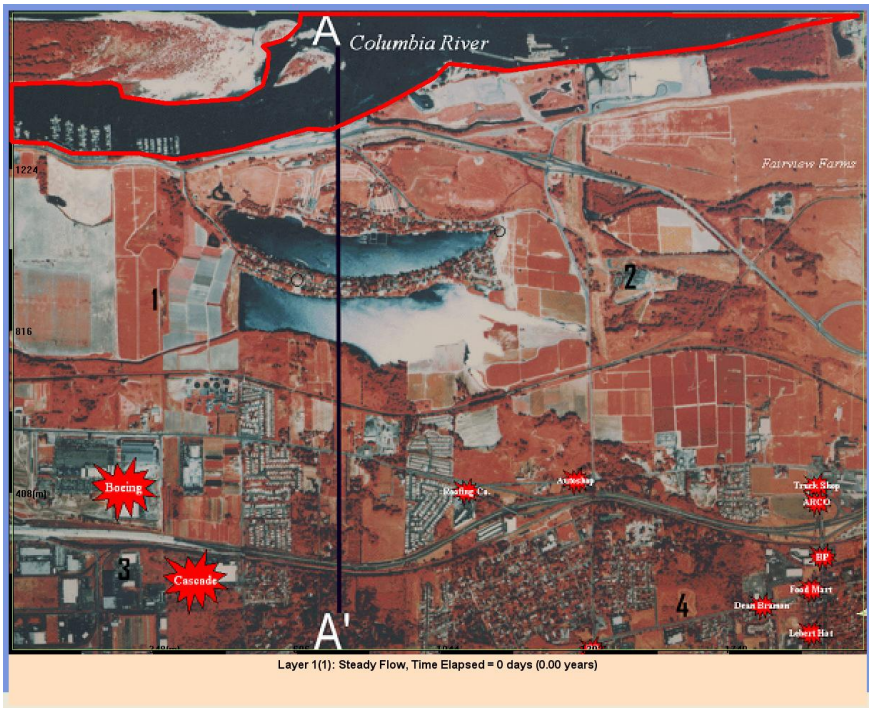
So far you have created the parent zone for the model, assigned the dimensions of the aquifer and the hydraulic conductivity of the aquifer. Now you will add Columbia River in the model.

Step 1: Click the 'Create New Arbitrary Zone and Assign Property' button. The cursor is now initialized to add zones in the Working Area

Step 2: In the Working Area, click the pointer on any edge of the feature you wish to define. [Columbia River for this example]

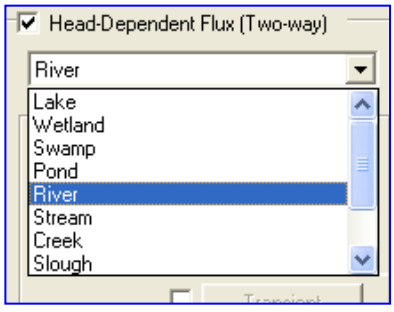
(*Note*: If you are not using a basemap, simply click anywhere to begin defining an arbitrary feature.)

Step 3: Click the mouse at another location on the edge of the feature. Move to another location and repeat, clicking as close to the exact tracing of the Columbia River as possible (for precision purposes). When you have finished, double-click the mouse. Try to keep the lines as close to coinciding with the edge of the feature as possible.



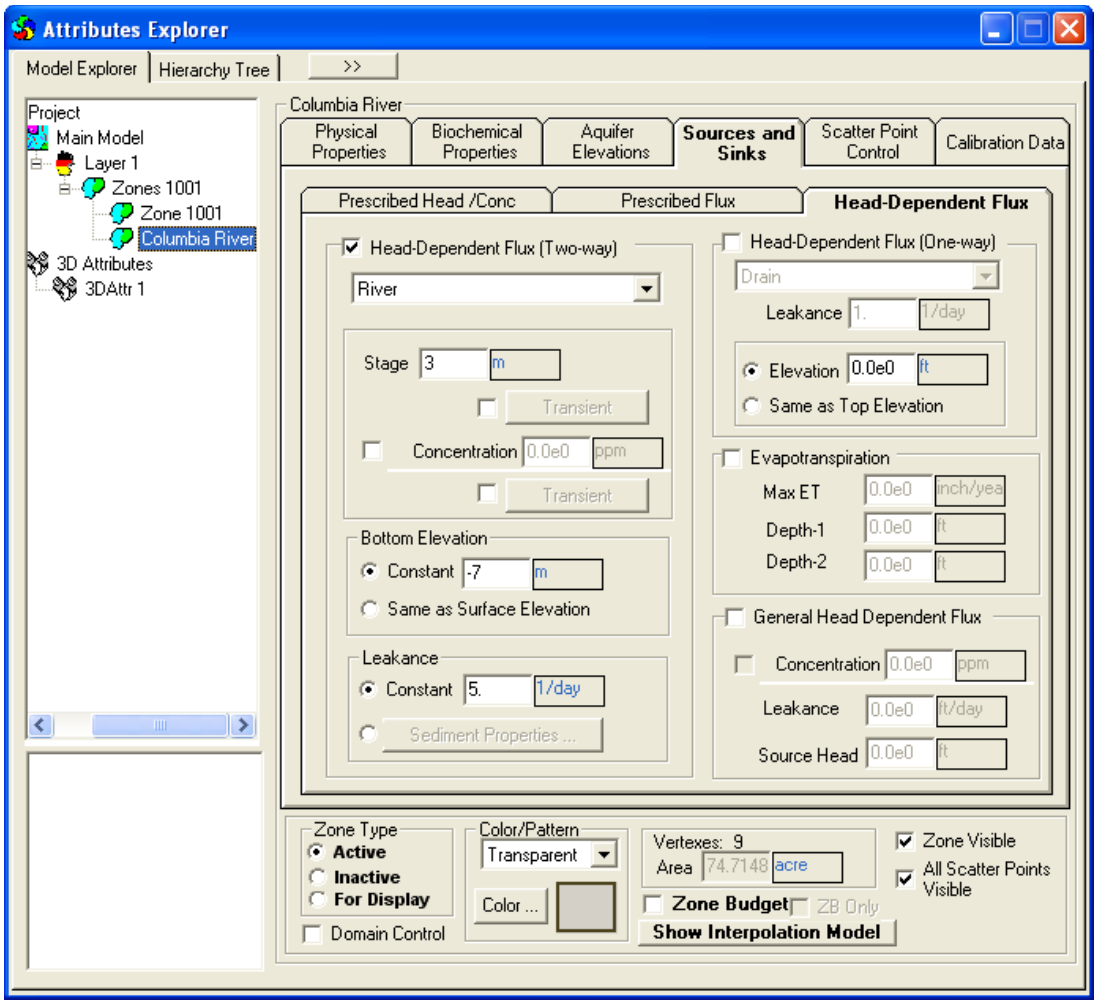
Step 4: Access the AE to assign attributes for the newly defined Columbia River Zone:

a) Click once on the default name 'Zone 1002' in the left hand pane and rename it as 'Columbia River'.
b) Click the 'Sources and Sinks' tab.
c) Select the type of source / sink that you would like to model. For this example, model the Columbia River as a Head Dependent Flux by selecting 'Head-Dependent Flux' tag. Check 'Head-Dependent Flux (Two-way)' area. Click on the arrow next to the top field in this area and select 'River' from the drop down menu (see *Note* after Step 5). Select unit [m] and type [3] in the 'Stage' field.



Step 5: Go to 'Bottom Elevation' area, check the radio button for 'Constant', select unit [m] and type in [-7] in the input field. By default, 'Leakance' value is 5 /day. Do not change it at this stage. Consult the IGW User's Manual for explanations of the types of sources / sinks.

(*Note*: All the names in the drop down menu in step 4(c) are for identification of features only. Selecting any of these names will not affect modeling result. By assigning different categories to different features in model, you will be able to differentiate them in water budget estimation. Water budget is explained in **Section 16.2**



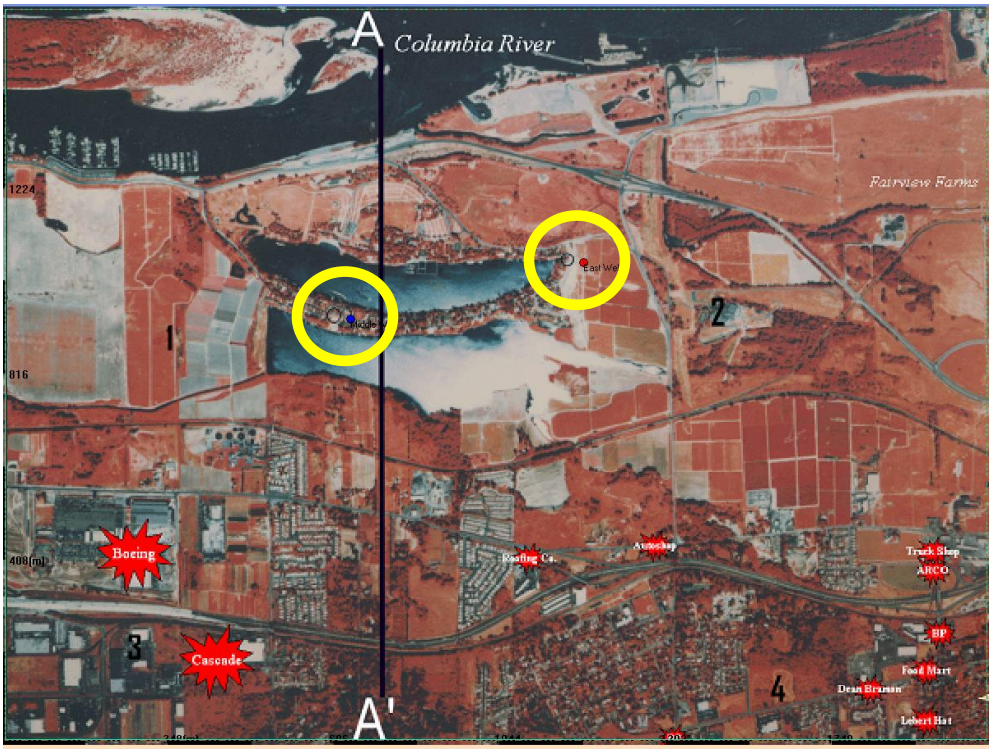
3.5. ADDING WELL FEATURES

A well is simply another type of source or sink, depending if it's a pumping well (sink) or injection well (source). The following steps illustrate how to add a pumping well to the model. Recall that in our conceptual model, we have two resident's wells pumping from the sandstone aquifer. To put the wells in the second model layer:

Step 2: Click the '[Add a New Well](#)' button, located at row 2, column 4 of the button palette. The cursor is now initialized to add well features to the model.

Step 3: Bring the cursor to the approximate location where a well is to be located (in this example, first, at a location between Blue Lake and Fairview Lake and then at the eastern end of Blue Lake) and click once. A well is added at that location, appearing as a red dot. Move the cursor to the next desired location and again click once. Another well is added at this location. Notice that the latter well now appears as red and the former one turns blue.

The locations of the two pumping wells just added into the model are highlighted with yellow circles below.



Step 4: Access the AE to assign attributes for the newly defined wells.

You will see two new wells in the 'Attribute Explore' pane of AE with default names.

Step 5: Edit well names as 'Middle Well' and 'East Well'.

Step 6: Select 'Middle Well'

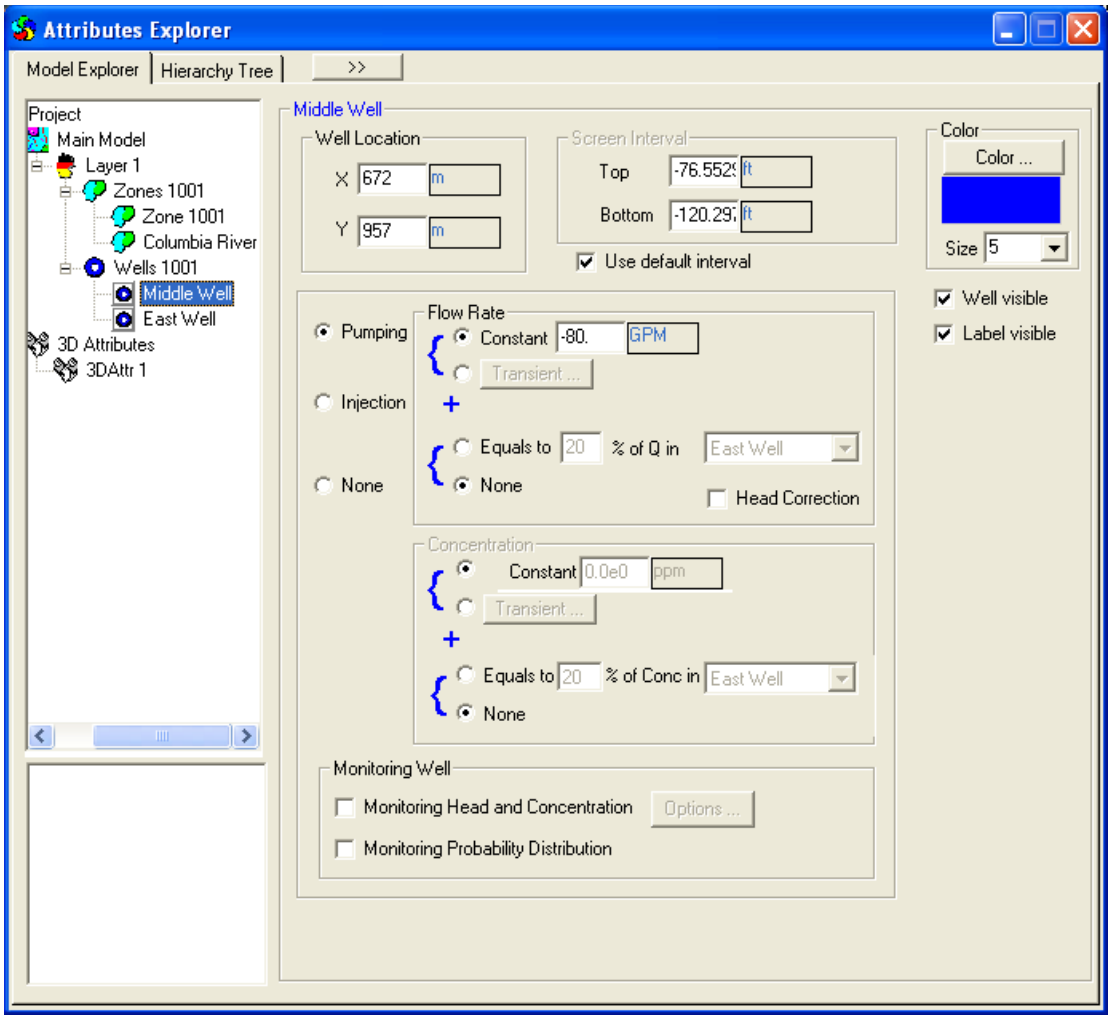
Step 7: In the 'Well Location Area' enter the exact location coordinates of the well. Choose unit to [m] and enter values [X = 673.0 ; Y = 957.0].

Step 8: Make sure that 'Pumping' is selected in the 'Well Type' area. (The other options are 'Injection' and 'None').

Step 9: In the 'Flow Rate' area, select 'Constant'.

Step 10: Change the unit field menu to the desired units [GPM]. For this example, we will use the standard GPM (gallons per minute).

Step 11: Enter a value in the 'Constant' field of [-80]. The (-) values indicate water being withdrawn from the well.



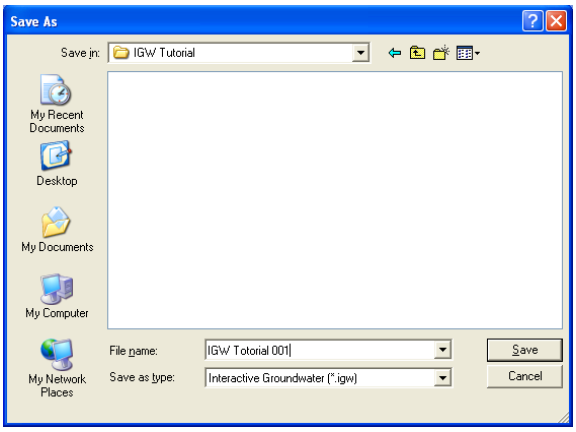
Step 12: Select 'East Well' in the left hand pane and repeat steps 7 to 11. [In Step 7, enter X=1213; Y=1066]

3.6. SAVING THE CONCEPTUAL MODEL

By now we have completed the basic model structure and added various sources and sinks in the model. It is always a good idea to save the model at this stage before discritizing and running it to prevent data loss and the time/effort spent to build the conceptual features. The model will be saved as a 'conceptual model'. You can close the model after it is saved and open in a later session of IGW. Data input and model features can be edited in any later session.

Step 1: Click on the Save Model button.

The 'Save As' window appears.



Step 2: Surf to the location you would like to save the file.

Step 3: Type in a name

Step 4: Click the 'Save' button.

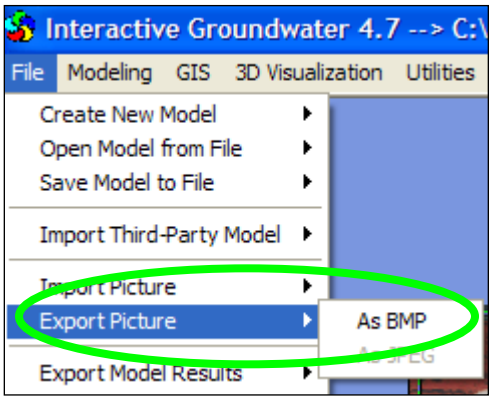
You may close the model now and open in a later session, or just continue working.

3.7. SAVING THE DISPLAY AS A PICTURE

It is sometimes desirable to save the program model screen for use in presentations or reports. This section will show you how to do that.

Step 1: Select the **File** menu.

Step 2: Select 'Export Picture' on this menu.



Step 3: Select the file type from the cascading sub-menu. *(Note: IGW currently only supports BMP.*

Step 4: Select a location and file name and press the 'Save' button.

The program saves the picture for your future use.

Pressing the 'Print Screen' button on the keyboard also saves the current screen in the Windows™ clipboard. You can subsequently edit the picture with a variety of graphics editing programs ranging from

the relatively simple 'Paint' included in Windows™, to powerful third-party software packages such as Adobe PhotoDeluxe™.

A simple example of this technique: given a screen in IGW you want to capture, press the 'Print Screen' key, located above the F12 button on your keyboard. Next, go into MS Paint and hit 'Ctrl-V', which will paste that image into the program. Then, use the cropping tool (row 1, column 2 of the MS Paint button palette) and select the area you wish to cut and use. Finally, once selected, go to 'Edit', then 'Copy To…', and select a location in which to save your file. This image is now ready to be imported into any program!

3.8. OPENING A SAVED MODEL

To open a saved model:

Step 1: Start IGW

Step 2: Click on Open a Model button.

Step 3: Browse to the saved model location, select the saved model, and click ok.

The saved model opens in IGW Working Area.

(*Note*: If you double click on the saved model in Windows Explorer, it will only open IGW with a blank working area. To open your model you have to follow steps 2 and 3.

CHAPTER 4: OBTAINING A SOLUTION

This chapter will walk you through the basic procedures for converting a conceptual model into a numerical model in IGW 4.7, and finding its solution.

4.1. DISCRETIZING THE MODEL

Discretizing the model turns the conceptual model into a numerical model that the computer can solve.

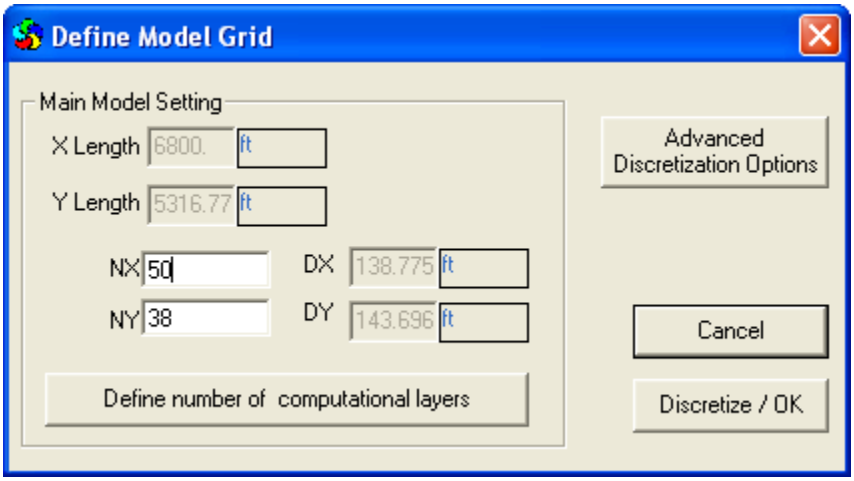
Step 1: Click the 'Map into Numerical Model (Incremental Discretize)' button to discretize the model, located at row 7, column 2 of the button palette. Use this step only if you do not want to change default settings.

Default setting is a model grid of 35 x 27 grid (not visible) that covers the entire Working Area.

Start from step 2 if you to set a specific grid size or grid characteristic.

Step 2: Click the 'Set Simulation Grid' button, located at row 7, column 1 of the button palette.

This brings up the 'Define Model Grid' window. Here you may enter the number of sections in the x-direction (NX). The computer then automatically calculates the number of cells in the y-direction (NY), and the cell dimensions DX and DY based on this number, along with dimensions of the model screen ('X Length' and 'Y Length'). A higher grid resolution yields a more accurate solution, but can dramatically increase computational time. The grid resolution is left to you, as it is very dependent on the problem being solved and the speed of the computer being used to solve it.



Step 3: Enter [50] in NX field. NY field will be automatically adjusted.

Step 5: Click 'Discretize/OK'.

Step 6: For any future adjustments or modifications to your model, always use the 'Map into Numerical Model' button. Doing this will only update the changes made to your model, versus having to re-discretize the entire model, which would take a lot of unnecessary time.

Once these steps have been completed, the model is now ready to solve.

4.2. SOLVING THE MODEL

Once the numerical model is ready, the next logical step is to solve it. (Continue only after you have discretized your conceptual model.)

Step 1: Click the 'Forward' button, located at row 9, column 4 of the palette.

By default, the model is solved at steady state.

At this stage, you have a complete working model. Now you can use this model to answer the questions raised by the community. You can also:

- View the model results across any cross section(s);
- Change the way the model outputs look like;
- Add contamination plumes/particles and do transport modeling;
- Change stresses in the model (e.g. change pumping/recharge rates) to test different scenarios; and,
- Add more complexity to the model.

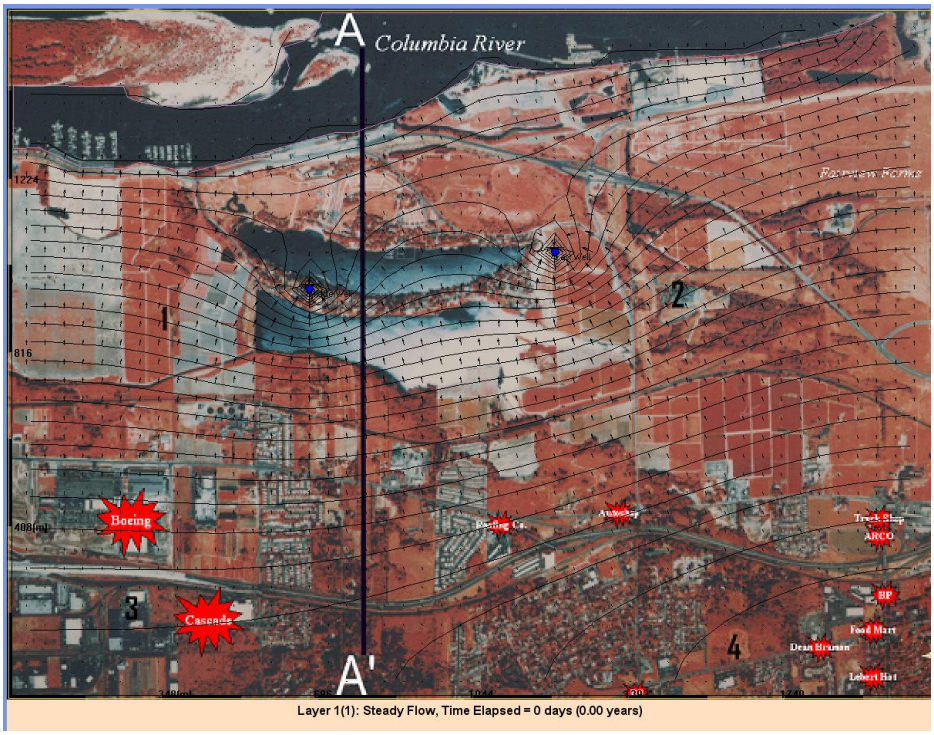
The next chapters will walk you through these procedures.

CHAPTER 5: VIEWING MODEL OUTPUTS

As soon as the model is solved, the model solution appears in the working area.

5.1. VIEWING HEAD AND VELOCITY FIELDS

Simulated groundwater heads appear in the working area as contours and groundwater flow velocity as arrows. The length of the arrows is proportional to the magnitude of the groundwater velocities. The following head and velocity fields indicate the general flow direction towards Columbia River and cones of depression formed around the pumping wells. This pattern of heads and velocity is consistent with our conceptual model of Figure 3.0.0..



5.2. DEFINING AND VIEWING CROSS-SECTIONS

A cross section cuts through all layers in a model, therefore it can be drawn in any layer and will show up in all model layers (although for now, you have only one layer in the model, but you will see that when more layers are added, the cross section is modified automatically) .

Cross-sections can be viewed in a non-linear fashion. Instead of just selecting a straight line section, you can simply keep defining the cross-section in any desired direction to include points that may be of interest. For this example, you will start the section at A, pass it through Middle Well and finish it at A' in the working area using the following steps:-

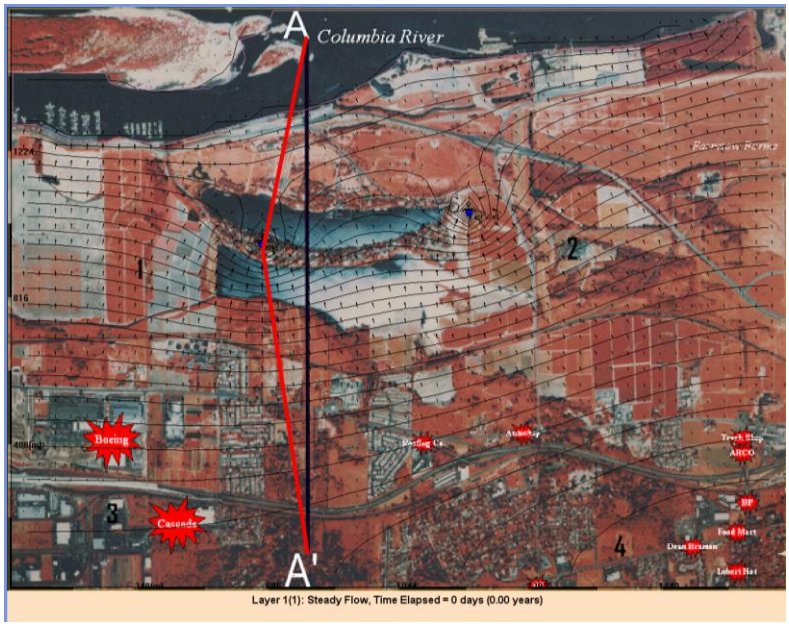
Step 1: Click the Define a cross section button.

The cursor is now initialized to add a cross-section in the working area.

Step 3: Click on a point where you would like to begin your profile view. For this example, start your cross-section at pre-defined cross-sectional view near the letter A, at the top of the view.

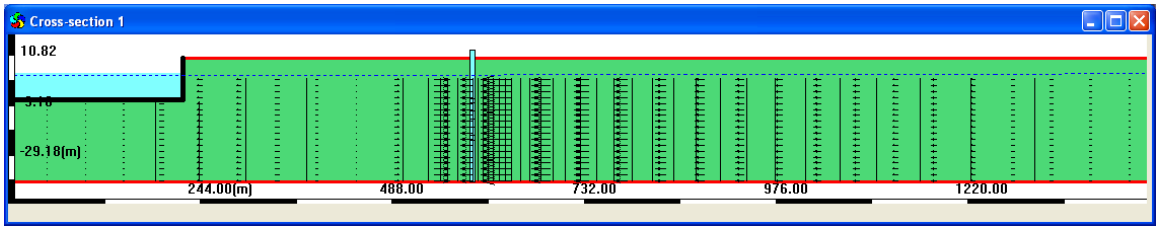
Step 4: Move the cursor to the Middle Well and click on the centre of the well. This will include the well in your cross section.

Step 3: Move the cursor to the point where you would like to stop your profile view and double-click the mouse. Do this on the pre-defined cross-sectional view near A', at the bottom of the view.



Plan view of the cross section so defined appears as bold red line in model area.

A window appears titled 'Cross-Section X' (where X is a number pre-determined by the IGW software).



The cross-section includes the stratigraphy, the velocity profile, a continuous head representation, and any model features that the cross-section intersects. The cross section above shows Columbia River and Middle Well. Blue dotted line shows the head profile, vertical black lines represent head contours and black arrows indicate magnitude and direction of flow velocity.

The cross-section is merely a display of what has already been solved for (it is not a separate model).

Currently, IGW does not indicate any three-dimensional attributes in the cross-section window; so if you define a complex cross-section keep in mind what it is you are actually viewing.

You can also view the contamination plumes and particle movement if the cross section intersects their path. When the model is running, the cross-section(s) are also updated with every visualization step.

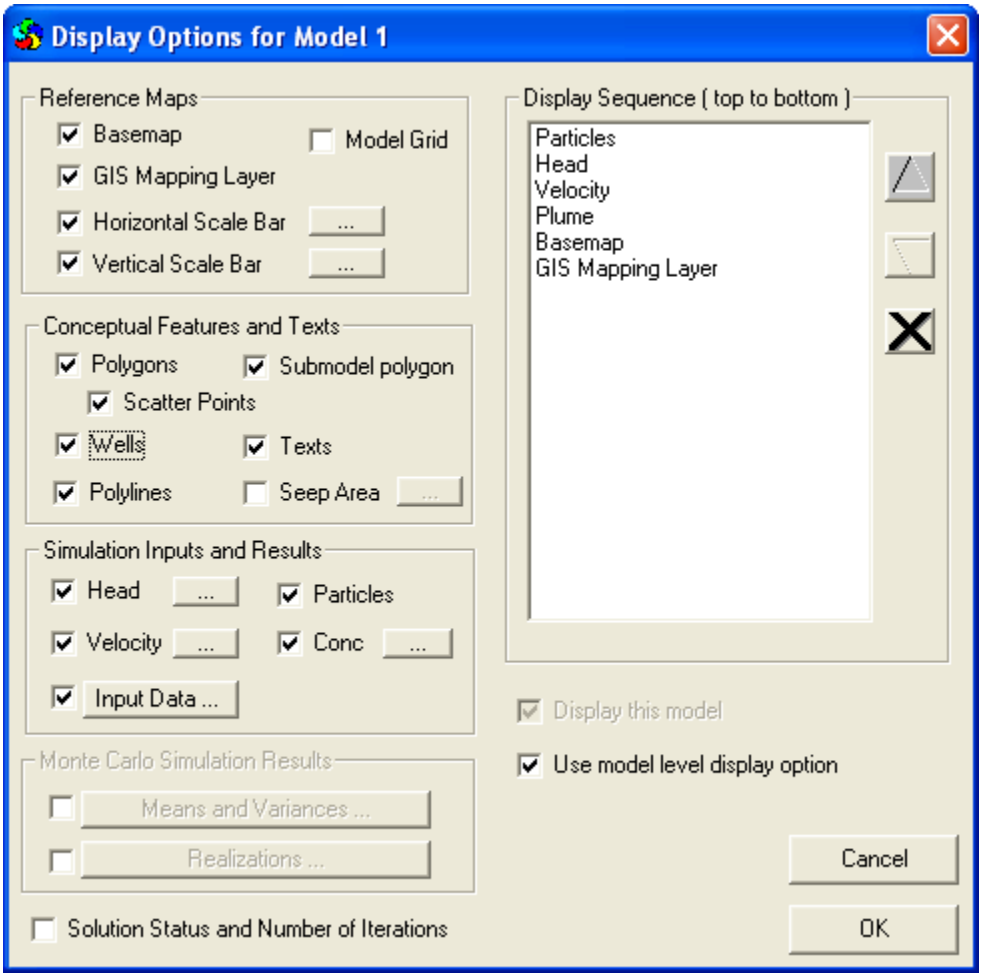
CHAPTER 6: CHANGING DISPLAY OPTIONS

Default settings for viewing the model output and cross sections may not always give the best visualization especially with a basemap in the screen. This chapter will walk you through the steps and tools available in IGW to change and optimize display settings in the model area for better visualization:

6.1. CHANGING DISPLAY IN MODEL AREA

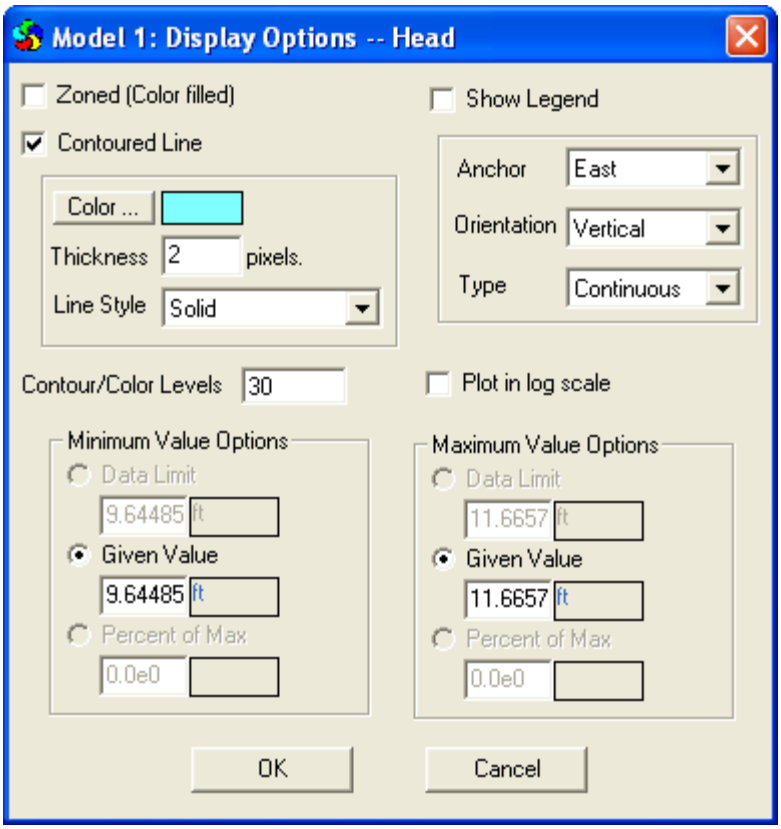
Step 1: Click the 'Set Drawing Options' button, located at row 8, column 1 of the button palette.

The 'Display Options for Model 1' window appears. Using options in this window, you can change the way the model may look. You can check or uncheck various boxes to select what items you want to see on the model. The right hand pane (Display Sequence) can be used to arrange various features in the model. You can explore these options as you go. For now, you will change the appearance of head contours and velocity fields in the 'Simulation Inputs and Results'.



Step 2: In the 'Simulation Inputs and Results' area, click the button ... to the right of 'Head'.

The 'Model 1: Display Options -- Head' window appears.



Step 3: Click the button

. The 'Color' window is now open.

Step 4: Click on the light blue color, firth in the top row, (or any other that you feel will be effective) then click 'OK'.

You return to the 'Model 1: Display Options -- Head' window and you can see that the color is now to the one you chose (light blue in this case).

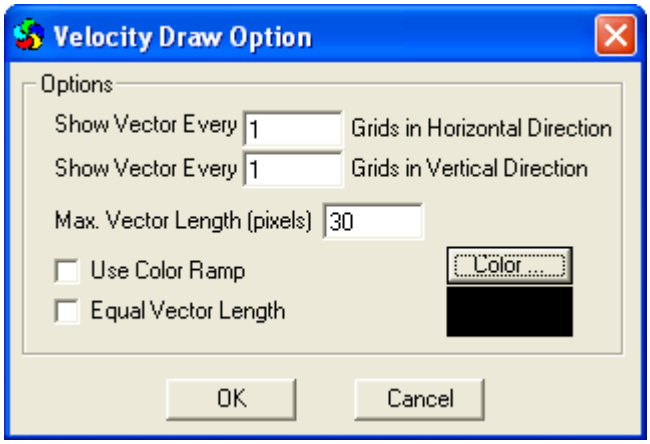
Step 5: In the 'Thickness' field, enter 2 (or any other that you feel will be effective).

Step 6: Click 'OK'.

You return to the 'Display Options for Model 1' window. You can explore more options on the 'Model 1: Display Options -- Head' window such as line style, legend, number of contours, maximum and minimum value for contours etc. etc.

(*Note*: If you change maximum or minimum values and/or number of contours, you have to uncheck the 'Use model level display options' box on 'Display Options for Model 1' window before clicking on any of ... buttons. If you don't uncheck 'Use model level display options', the changes will not take effect).

Step 7: In the 'Simulations Inputs and Result' area, click the button ... to the right of 'Velocity'. The 'Velocity Draw Option' window appears.



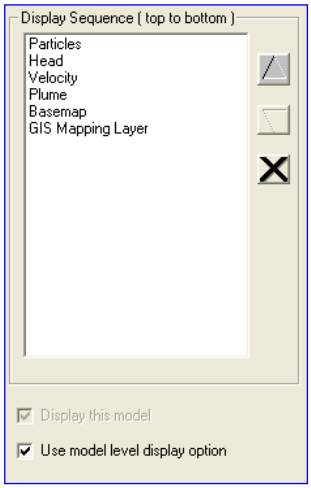
Step 8: Click the Color ... button. The 'Color' window is now open.

Step 9: Click a yellow color (or any other that you feel will be effective), then click 'OK'.

You return to the 'Velocity Draw Option' window. You can explore other options in this window.

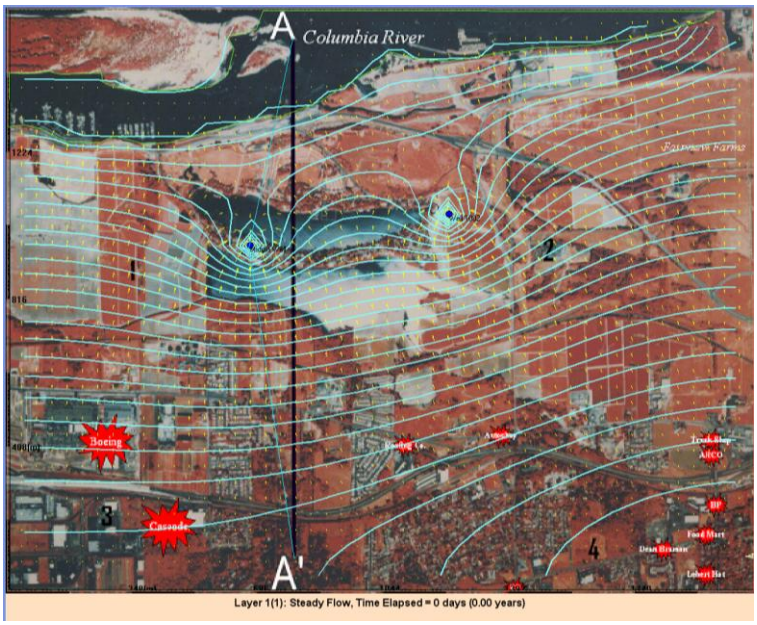
Step 10: Check the 'Equal Vector Length' box, set 'Max. Vector Length (pixels)' to [20] and click the 'OK' button.

You return to 'Display Options for Model 1' window. In the 'Display Options for Model 1' window you can see the 'Display Sequence (top to bottom)' area. This area shows the order of display for the model screen (i.e., the last item on the list is displayed below the other items). In some cases it may be helpful to adjust these settings or to remove non-essential elements. It is left to you to experiment with these settings



Step 11: Click the 'OK' button.

Notice that by giving a color to the contours which more contrasting with the background gives a better visualization of head distribution in the model. Moreover, by drawing the velocity vectors of equal length, you can better visualize the flow filed in Layer 1, which was not very prominent in default display settings. Display settings are universal and apply to all model layers.



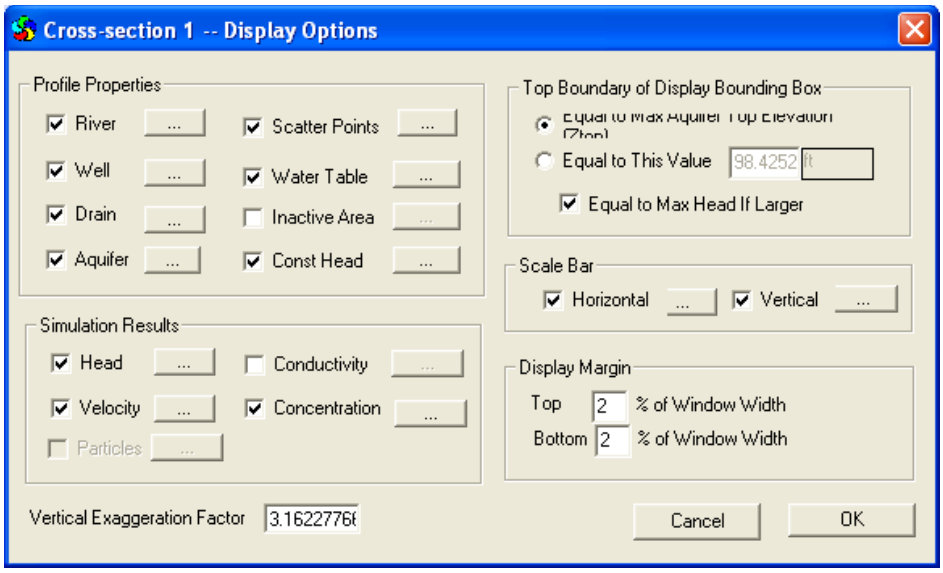
You return to the model screen and the drawing updates have taken effect. If the settings have not taken effect (or any time you wish to refresh the IGW screen), simply click the refresh button

6.2. CHANGING DISPLAY FOR CROSS-SECTION

You can change the appearance of the cross section. To change appearance:

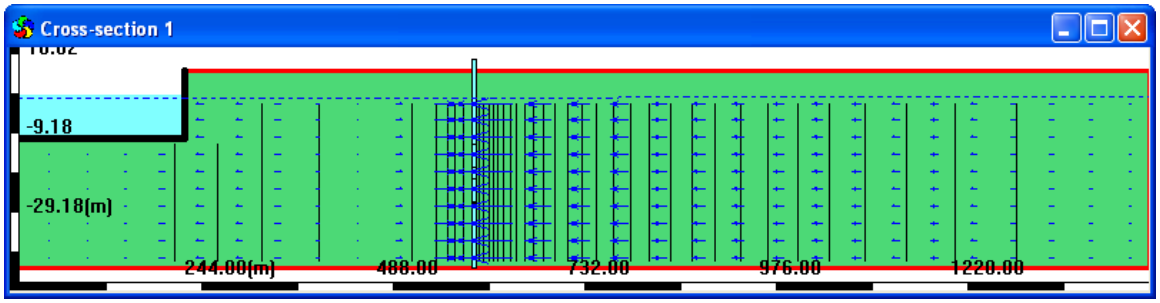
Step 1: Right click anywhere in the cross section window, a menu pops up.

Step 2: Click on the 'Draw Option' from this menu. 'Cross-Section X -- Display Options' windows pops up.



From this window you can explore the display options for each item by clicking on ... button in front of the item. For 'Head' and 'Velocity' options the button will lead you to pop up windows where you can chose the color and density of your choice for the head contours and velocity arrows. You can change the 'Vertical Exaggeration Factor' by changing the number in the field (bottom left of this window).

Using a vertical exaggeration factor of 5 and changing head and velocity appearance, the cross section is redrawn below.

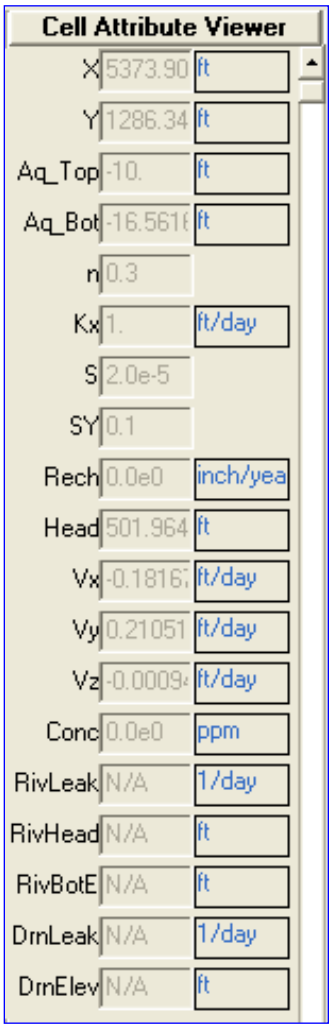


CHAPTER 7: EXPLORING THE CURSOR ACTIVATED TABLE

This short chapter introduces you to the Cursor Activated Table (CAT), mentioned in **Section 2.2**. CAT allows you not only to see the model output at any given point in the model where cursor is placed, but can also give you any desired input data/attribute at that location.

7.1. THE CAT INTERFACE

As mentioned earlier, the CAT occupies a major portion of the right side of the main window.



If you move the cursor around the working area, you will notice the respective values change to display the values that exist at that exact location in the model. This is done only in the active layer. [Select Layer 1 and move the cursor in the model area and notice the display in 'Rech' field. You will notice it is [10 inch/yr] every where but turns to [0.0e0 inch/yr] as soon as the cursor enters a lake or a river polygon]

You can also change the units for particular variables in the CAT. This is accomplished by clicking on the unit box of the variable you wish to change (as we have done in previous steps inside the AE), and then simply clicking until you have selected the desired units. IGW will now display these units.

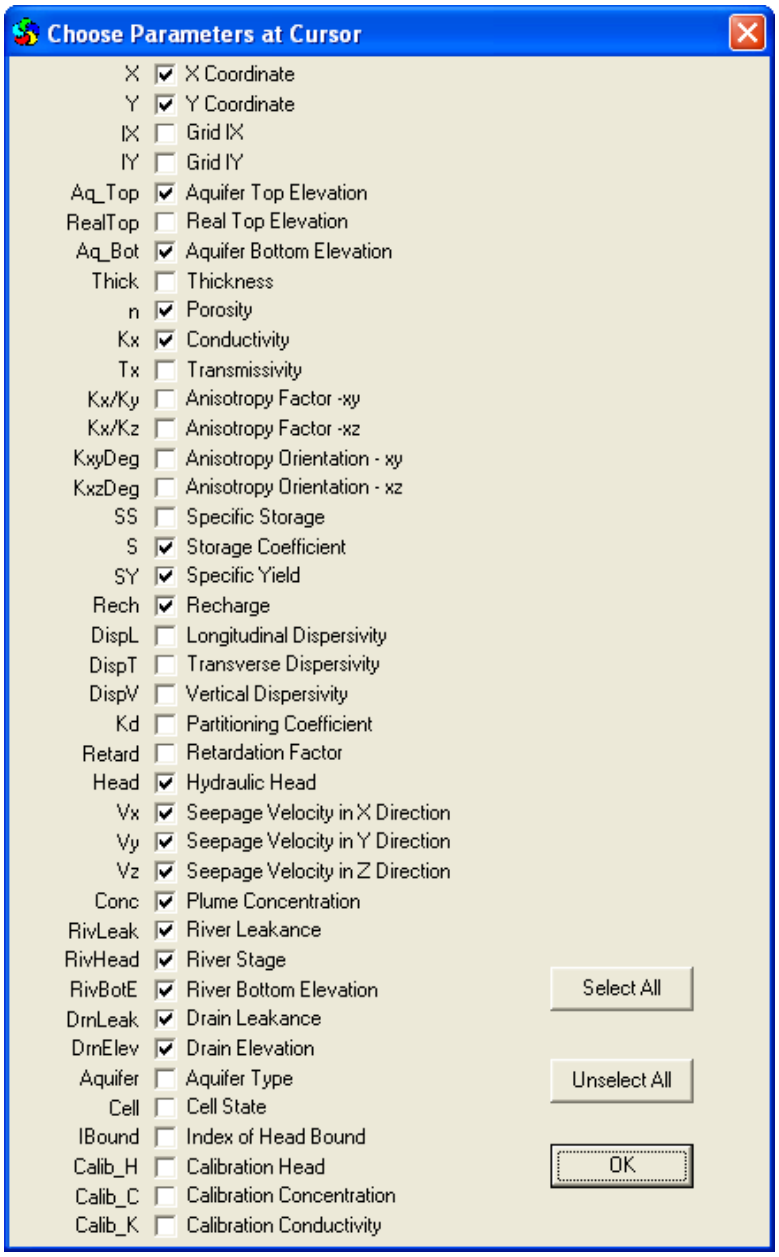
Also, there may be more variables in the CAT than are visible at a particular time. To view the hidden variables, simply use the scroll bar on the far right to access them.

7.2. ADDING/REMOVING FIELDS IN CAT

The values you see in the CAT are set by default, but you can change these fields as follows:

Step 1: Click on the 'Cell Attribute Viewer' button.

The 'Choose Parameters at Cursor' window appears. From here you can select which parameters you would like displayed in the CAT. The field names are also fully defined in this window. This is useful as the field names displayed in the CAT are truncated.



Step 2: Check/uncheck the fields that you want/don't want in the CAT.

Step 2: Click the 'OK' button to close the window.

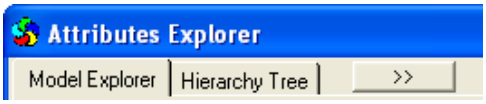
CHAPTER 8: EXPLORING THE ATTRIBUTE EXPLORER WINDOW

This short chapter will introduce you in greater detail to the Attribute Explorer window (AE). Previous versions of IGW refer to this tool as the Attribute Input and Model Explorer window (AIME). Version 4.7 shortens this to the Attribute Explorer (AE). You have been using the AE to input values for the features that you have defined in the previous chapters. Here you will explore some extra features of the AE.

8.1. THE ATTRIBUTE EXPLORER WINDOW

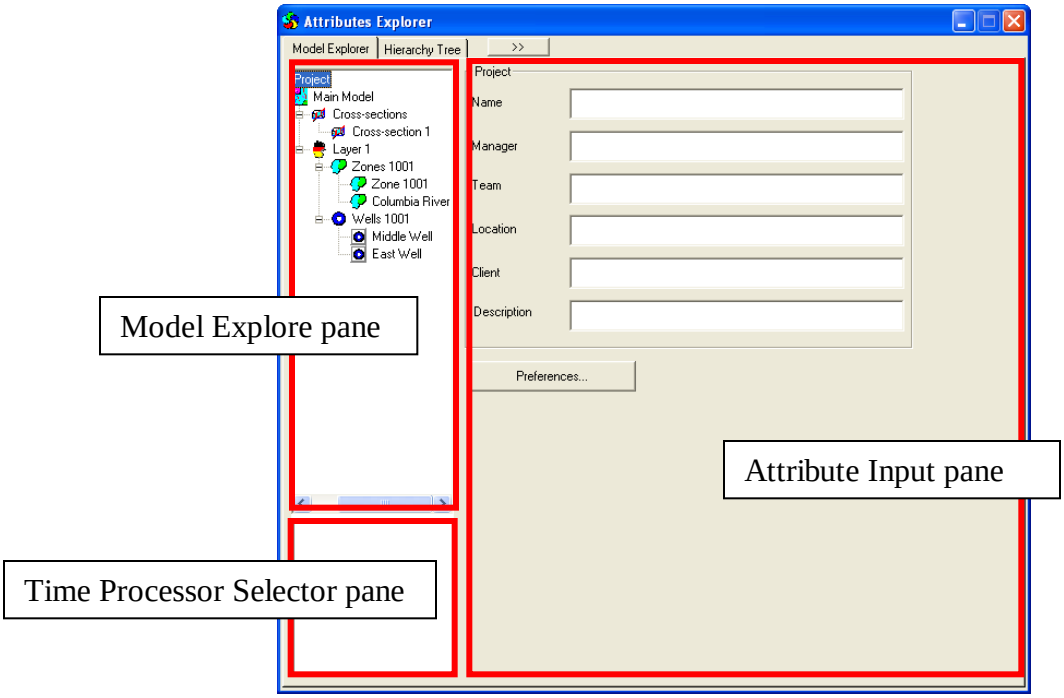
Bring the AE to a visible location.

You will notice there are two tabs in the 'Attributes Explorer' window, i.e., '**Model Explorer**' tab and '**Hierarchy Tree**' tab. The Hierarchy Tree tab is not yet functional in IGW 4.7. Net to the tabs is 'double arrow' button. You can use this button to toggle between narrow or wide display in the left pane (called the Model Explore pane) of the AE window. Wide display may be useful to see full text and hierarchy structure of the model.



8.2. MODEL EXPLORER TAB

'**Model Explorer**' tab is divided into three panes. The top left pane is the **Model Explore** pane which gives the hierarchical visualization of the model's components. The wider pane to the right is called **Attribute Input** pane. The structure of this pane changes depending on the hierarchical level and type of model component selected in the Model Explore pane. In the figure below, "Project" is selected in the hierarchical visualization pane. The input fields relating to 'Project' are displayed in Attribute Input pane. We shall see different attribute input fields as we shall deal with different components in the model.



The lower left pane is the Time Process Selector (**TPS**) which is seen empty in the figure above. From the TPS you can open a separate window for any previously defined time process (mass balance, mass flux, well head, well concentration, etc.) and view the results as the model solution proceeds through time. As we currently do not have any of these functions incorporated into our example, you can see that this pane is an empty box in the lower left corner. It is more intuitive to explore the TPS with a working model, so the TPS discussion is continued in **Section 11.3**.

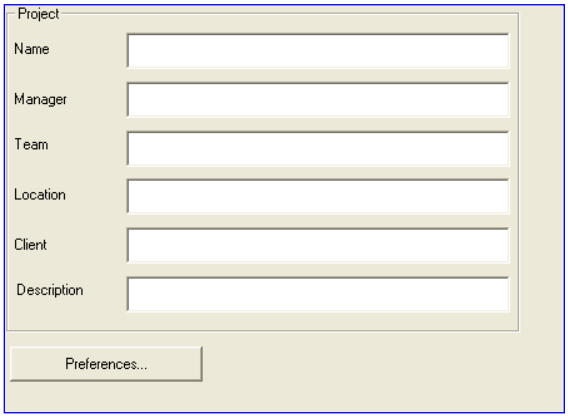
Unlike previous versions of IGW, the 'Apply' button no longer exists: **IGW 4.7 automatically updates your data the moment you input it into the AE!**

8.3. ATTRIBUTE INPUT PANE

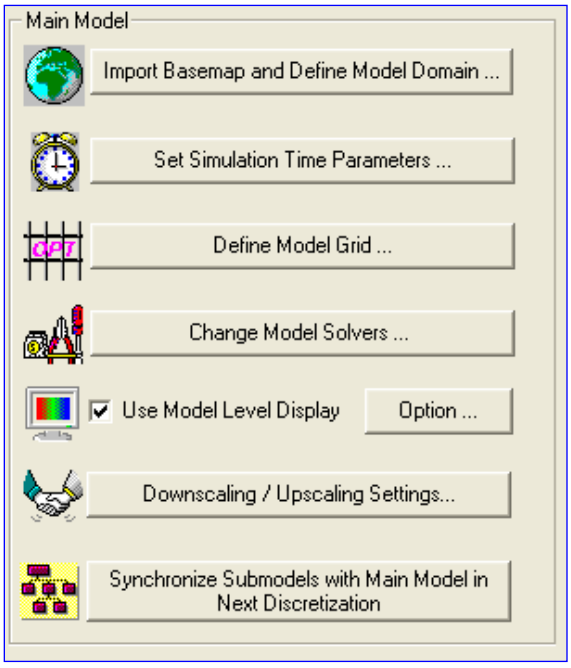
The fields and structure of the attribute input pane changes with respect to the type of item selected in the model explore pane. You can explore this feature as follows:

Step 1: Click on 'Project' in Model Explore pane of 'Model Explorer' tab. The Attribute Input pane displays input field for general project attributes.

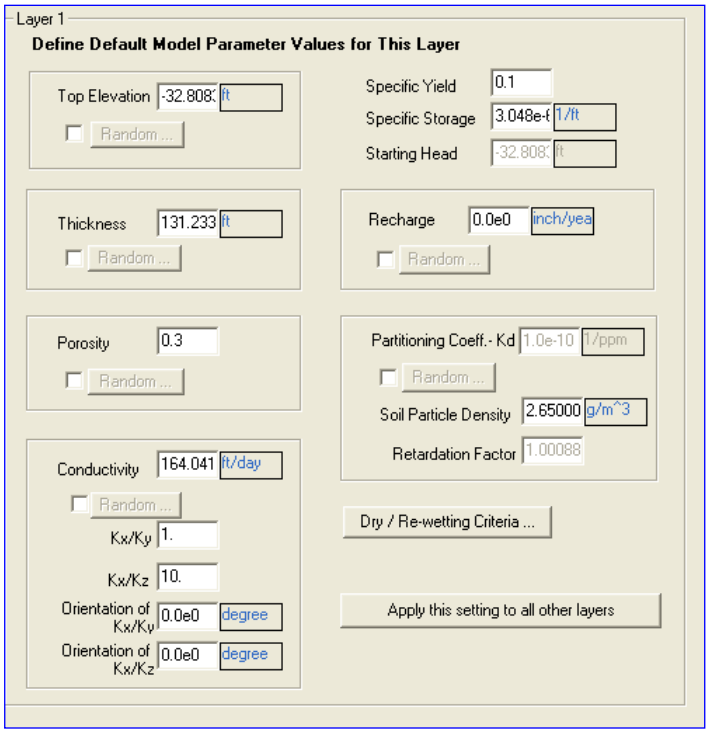
Step 2: Enter desired information (name the project whatever you wish to call it, etc).



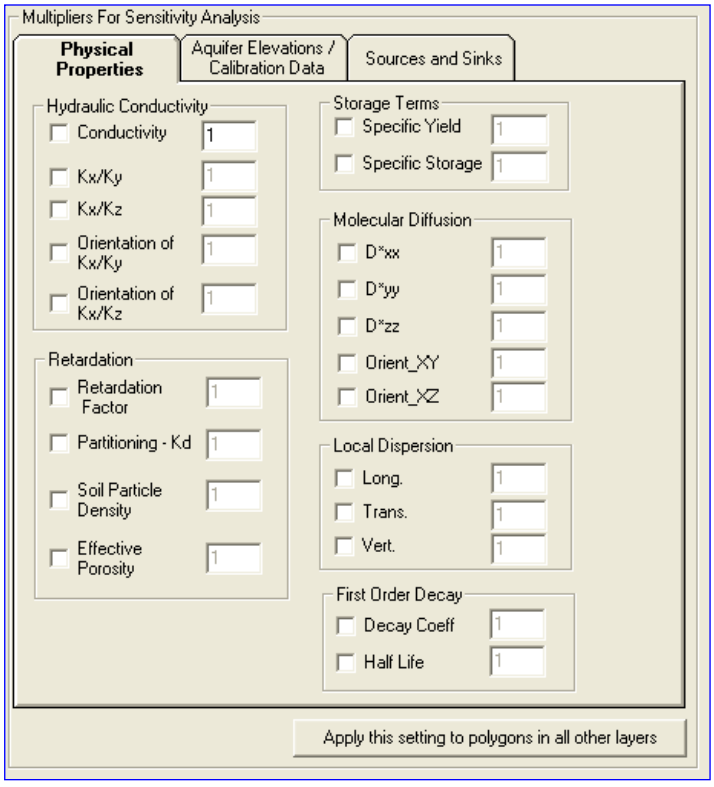
Step 3: Click on 'Main Model' in the Model Explore pane, located just below 'Project'. The Attribute Input pane now displays a collection of buttons (also available on the button palette) that can be selected to alter attributes for the model. Options are also available here for physical / chemical process parameters and particle visualization settings.


Main Model
Import Basemap and Define Model Domain ...
Set Simulation Time Parameters ...
Define Model Grid ...
Change Model Solvers ...
Use Model Level Display
Option ...
Downscaling / Upscaling Settings...
Synchronize Submodels with Main Model in Next Discretization

Step 4: Click on 'Layer 1', located below 'Main Model' in the Model Explore pane. The Attribute Input pane now displays a collection of fields in which you may enter specific data for this layer. Most of these values are accessible at other points within the software.



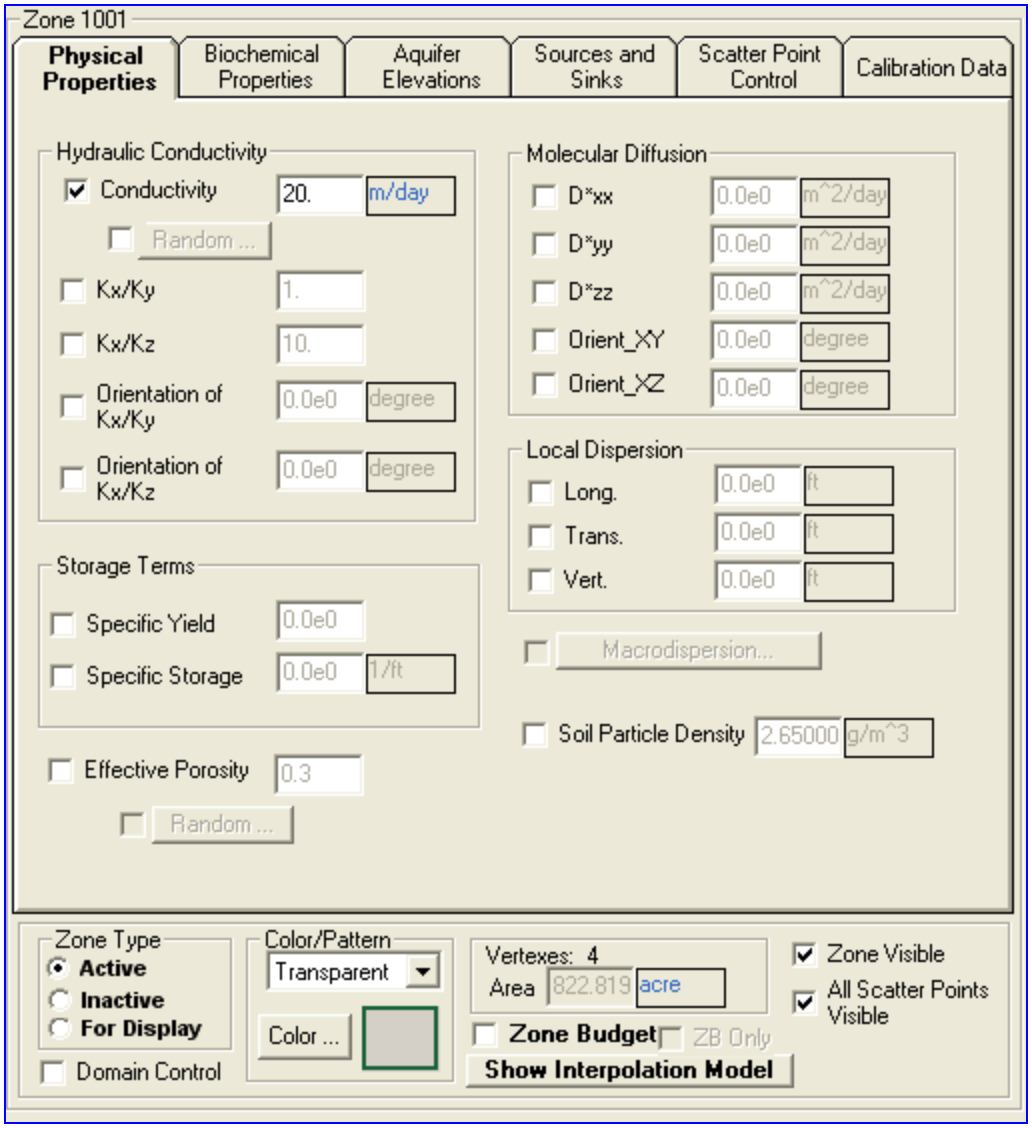
Step 5: Click on 'Zones XXXX' in the Model Explore pane (where XXXX is a number assigned by the software. In our example, this is 'Zones 1001').



In previous versions of IGW nothing appeared in the Attribute Input pane, as it was just a placeholder. Now this is the 'Multipliers for Sensitivity Analysis' region, which can be applied to Physical Properties, Aquifer Elevations/Calibration Data, and Sources and Sinks

A more detailed discussion of this feature and its usefulness is found in the User's Manual.

Step 6: Click on one of the zones that are in the 'Zones XXXX' group (for our example, click on 'Zone 1001'). The Attribute Input pane now displays a multi-layered section for entering information about this particular zone. Notice this is where you entered data for the parent zone and water body sources / sinks you defined in previous sections.



The Attribute Input pane will be will be different for every type of feature that you can define in IGW 4.7, including (but not limited to): wells, particle zones, profile models, sub-models, line features, and 3D-attribute zones. As you work through the tutorial, observe the AE and take notice of the new features that become available as your model increases in complexity.

8.4. EDITING FEATURES IN MODEL EXPLORE PANE

By left-clicking once on the feature in the Model Explore pane, you can re-name it whatever you want.

By right-clicking on the features and groups in the pane you can perform such functions as deleting them, viewing attributes, and reading in scatter points .

CHAPTER 9: PARTICLE TRACKING

This chapter will show you how to track contaminant particles in the model.

9.1. FORWARD PARTICLE TRACKING

The following steps will show how to track a collection of discrete contaminant particles.

Step 1: Click the 'Add Particles Inside a Polygon' button.

The cursor is now initialized to add particles.

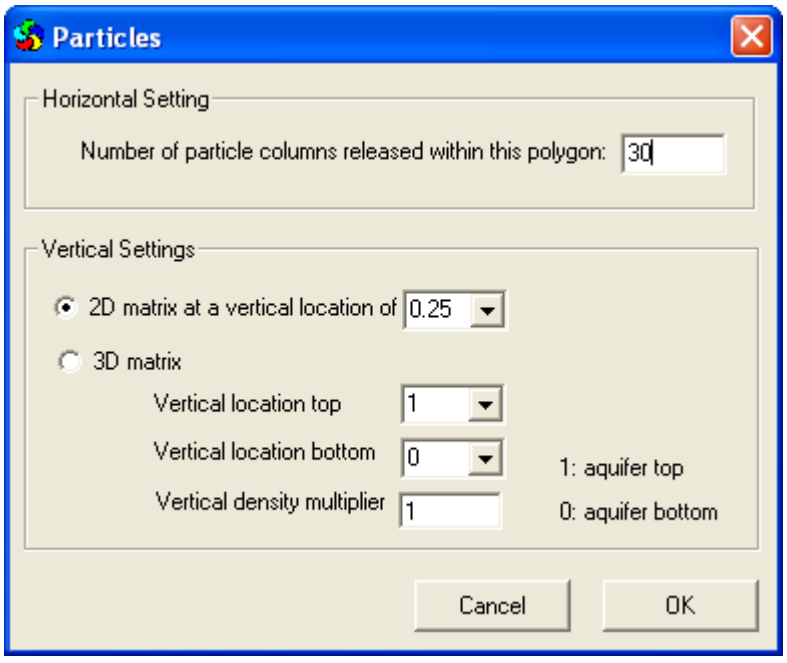
Step 2: Trace an area [Boeing factory location on the basemap], using the same methodology as used to define zones **(Section 3.3)** to define contamination plumes. This is shown on the map below as an enclosed red area; the green circle shows you its location.

Step 3: As soon as you finish the zone in which to put the particles, you will be prompted with the 'Particles' window. Here you should define:

a) How many columns of particles you want to include in your area (number proportional to concentration; for this example use a value of [30])

b) The vertical location of the specified layer that the plane of particles will be released into.

Notice that you have two options: 2D matrix and 3D matrix. You will use a 2D matrix in this example. Notice the drop-down arrow next to the input field for 2D matrix. The numbers range from 1 to 0 – 1 corresponding to top of the layer and 0 to the bottom. You will select 0.25 for this example which places the plane of particles at the bottom quarter of the layer.

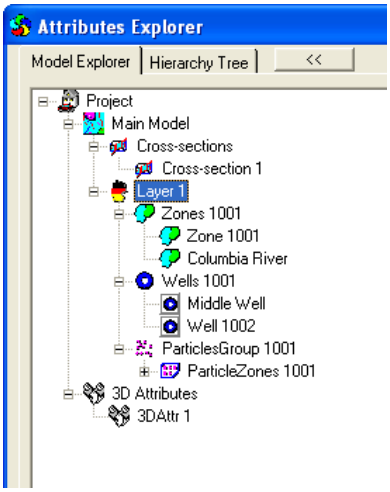


c) Enter values in these fields, and then click the 'OK' button.

Step 5: Repeat **Steps 3-4** for any other particle zones. [Do this for Cascade site.]

You can change the particle display settings (color and size of the particles) by going though the following steps:

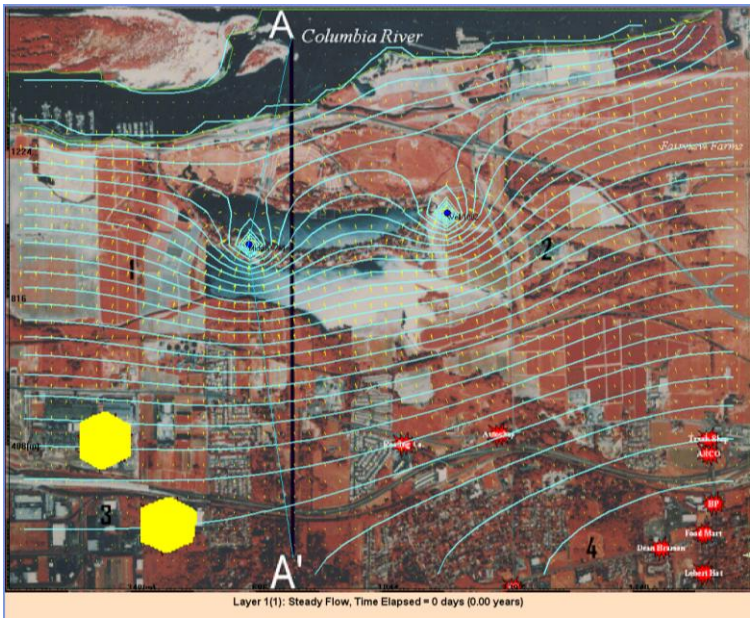
Step 1: Access the AE. In the model explore pane you will see 'ParticleZones 1001'. Click at the '+' sign next to the 'ParticleZones 1001'. The two particle zones you just created will open up.



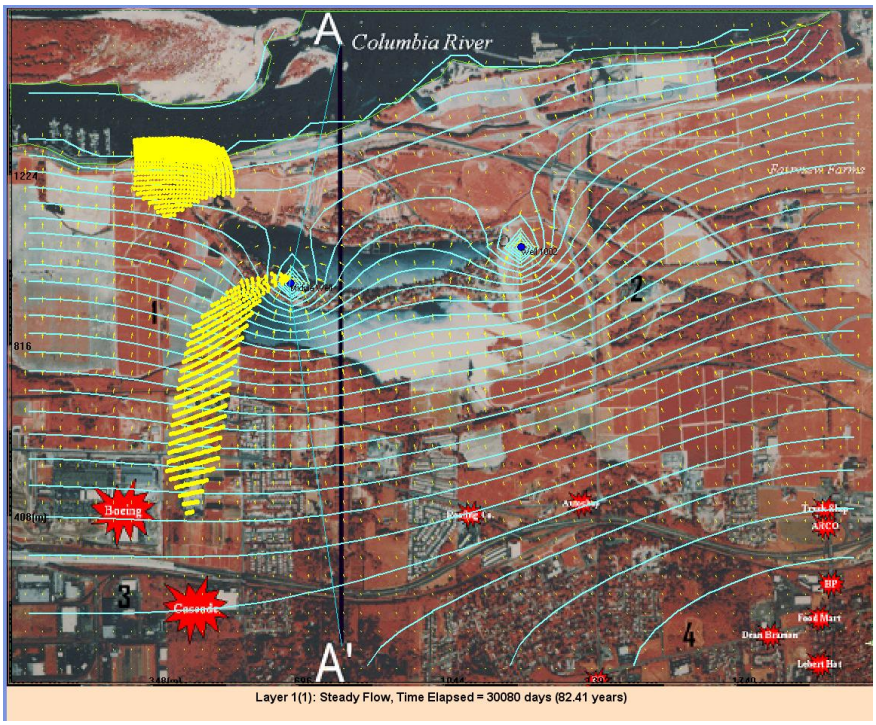
Step 2: Click on the first zone (for Boeing). Go to 'Display Options' area in the attribute input pane and select particle size [5] and color [yellow].

Step 3: Repeat above for Cascade site.

Your contamination fields are now defined in the model screen. In the following view, you see both contaminant particle "clouds" as yellow dots that are above the respective starting locations.



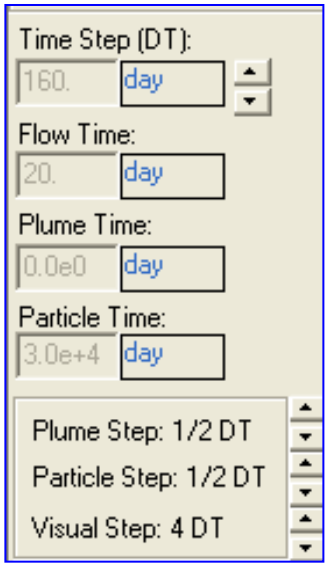
Step 4: Click the 'Forward' button to solve the model. (*Note*: Unlike all other model features, you do not need to discretize the model after adding particle features.)



Notice that the contaminant particles are following the velocity gradients, as expected. Also note WAAD for the time elapsed in the simulation. Here, the model suggests that Boeing contamination will end up in Columbia River after 82 years, but Cascade plume will be captured by the Middle Well.

Notice that with the particles present, the model solution now continuously updates at the time step indicated at the lower left of the main screen. You may need to adjust this time step to see any significant changes from one time-step to the next.

You can use SATDI interface to speed up the simulation by changing the Visual Step to a higher number. This instructs the software to redraw the simulation at fewer time-steps. At any time during the simulation, you can use Pause button to stop the simulation, reset (or initialize) the particles, change the time step, reset flow clock or reset particle clock. SATDI also provides the current values of these attributes as seen below. The numbers in SADTI area correspond to flow simulation at 30,000 days (82.4 years).



Note: These variables are highly dependent on the problem being solved and the speed/capacity of the computer running the simulation. Therefore, this tutorial leaves it to the reader to optimize these values. [Setting DT to 160 and using a visual step of 4 DT will give you good visualization results.]

Step 7: Stop the simulation by clicking the 'Pause' button

9.2. BACKWARD PARTICLE TRACKING

The following steps show you how to backward track a plume of contaminants. This is useful if you are trying to determine where some pollution came from.

Step 1: Click the 'Backward Particle Tracking' button

Watch the particles moving backwards and the time elapsed clock counting towards zero and going into the (-) numbers.

Step 2: Stop the simulation by clicking the 'Pause' button

9.3. INITIALIZING THE PARTICLES

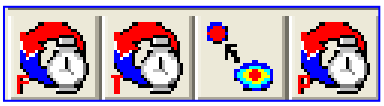
Step 1: Click the 'Initializing Particles' button to return the particles to their initial location, i.e. erasing the particles off the screen and returning them to their original starting position in the model

9.4. INITIALIZING THE CLOCKS

Step 1: Click the 'Reset Particle Clock' button to reset the clock

The time elapsed clock in the WAAD returns to 0 days.

Note: There are three similar looking buttons on the Button Palette in the 10th row. Each of these buttons is to reset the simulation time clock. The first one from left is to reset flow simulation clock, the second one to reset concentration simulation (transport) clock and the fourth one to reset particle simulation clock. Note the letters F, T and P in the lower left corners on the respective buttons to differentiate them for each other.



Until this point you have created a simple model. You learnt to add features into the model and assign/edit features' attributes. You have learnt to create cross sectional profiles and change display options both for model area and cross sections. You have also learnt to do particle tracking and use this feature to answer the question as to whether or not the contamination in the area can affect the pumping wells.

CHAPTER 10: ADDING COPLEXITY TO THE MODEL

The principal of parsimony is important to develop a model. It is usually helpful to start with most essential/basic elements of the model; then add more complexity as the model develops. IGW allows you to add complexity to your model in an incremental manner. IGW also allows you to visualize the impact of every change on the model predictions. In the following chapters, you will learn to add complexity to the basic model that you have created. You will:

- Add additional model layers and computational layers to represent the aquifer in more details;
- Add more features and elements in the model which were ignored on the assumption of being less significant;
- Modify aquifer dimensions with more detailed data on aquifer elevations;
- Add heterogeneity to aquifer parameters;
- Add contamination plumes; and,
- Add dispersivity in transport modeling.

As you will add more complexity to the model, you will also learn more features of IGW for visualization of results. You will learn how to visualize:

- Complex cross sections;
- Different types of 3D sections;
- Water budgets;
- Mass balance; and,
- Break though curves

CHAPTER 11: ADDING LAYERS

This section describes how you can add model (geological) layers and/or computational layers in a model. Each geological layer has its own properties, sources and sinks and boundary conditions. Computational layers can be added within the geological layers. All computational layers within a geological layer have the same properties and boundary conditions. Sources and sinks within the computational layers are based on their elevations (e.g. elevations of the well screens).

There are two ways to add a layer in IGW. The first is to access the **File** menu, then the **Create New Model** sub-menu, then select **Layer**. This creates a new layer below the original layer that has no features defined in it.

The second method is beneficial if there is a zone in the current layer that would be useful to import into the new layer. [This is the case as we will be using the same boundaries, thus the same parent zone, as the first layer. However the boundary justifications are different: the north boundary is a constant head due to the river, the south boundary is a no flow boundary because the aquifer pinches off and the east and west boundaries are no flow boundaries because of the regional flow direction.]

11.1. ADDING A NEW MODEL LAYER

First, make sure that the desired zone is selected by verifying the active red boundary around the zone in question [i.e. the basemap boundary or 'Zone 1001'].

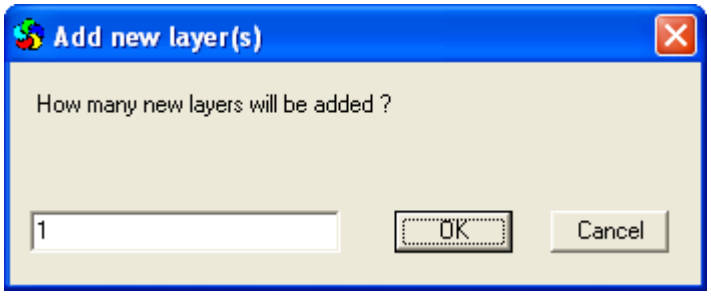
Step 1: Click the 'Select a Zone' button. The cursor is now initialized to select a zone in the Working Area.

Step 2: Click the mouse anywhere within the desired zone [Zone 1001]. The zone should now be highlighted with red borders (if it wasn't already).

(*Note*: if there are sub zones inside a bigger zone, then clicking in any sub zone will only select the sub zone. In this example, in order to select the parent zone, you should click the mouse inside the parent zone but outside the sub zones representing the lakes and the river)

Step 3: Press 'Alt+L' on the keyboard.

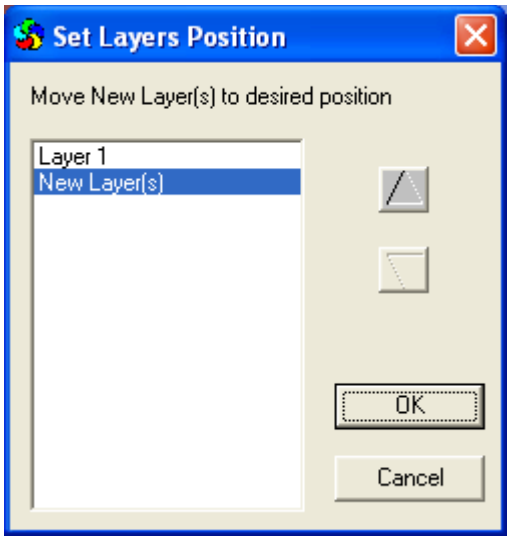
The 'Add new layer(s)' window appears.



Step 4: Enter the desired number of layers to add. In this example you will only add 1 layer.

Step 5: Click the 'OK' button.

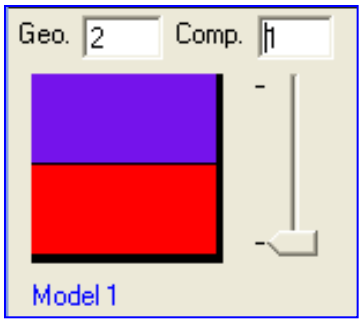
'Set Layers Position' window pops up. Here you can move the 'New Layer(s)' above or below the existing layer(s) by using the arrow keys. For this example, the new layer will go below the existing 'Layer 1'.



Step 5: Move the 'New Layer(s)' to the desired location using arrow keys and Click the 'OK'. In this model you are adding a New Layer below Layer 1.

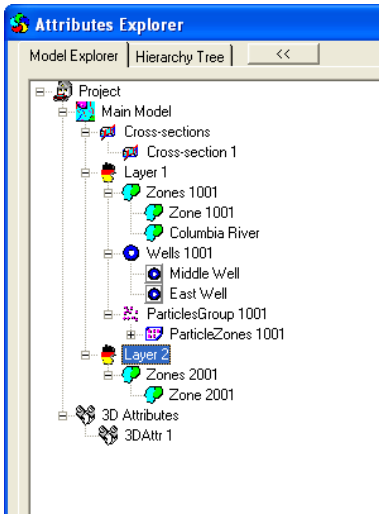
The software now adds the selected number of layers. A copy of the selected zone now exists in (each) new layer.

Notice that the Layer Selector now reflects the existence of the new layer(s). You can use the sliding arrow to switch between the layers shown in the Working Area.



Step 6: Access the AE.

Notice that the left-hand pane in AE now indicates the existence of the new layer(s) (and the zone that was copied to it). IGW will assign default layer and zone names. [For consistency and cross reference, keep the default layer names i.e, 'Layer 2' and 'Zone 2001']. You can edit the names by clicking once on the name in AE explorer pane and then retyping the new name.



11.2. EDITING LAYER ATTRIBUTES

The copied zone in the newly created layer needs to be modified, because its properties are identical to the layer it was copied from.

IGW allows you add, edit, or delete attributes of any layer in model. You can edit layer attributes such a elevations, hydraulic properties and stresses. The following procedure will show you how to edit layer elevations and other properties.

For the simplified conceptual model of Figure 3.0.0, you lumped aquifer properties and modeling features in one model layer. You now have a 2-layer model to represent more complex conceptual model of Figure 1.2.2. To re-assign aquifer elevations and different hydraulic conductivities to each model layer follow these steps:

Step 1: Use the Layer Selector sliding button to go to Layer 1.

Step 2: Click the 'Select a Zone' button and then click in model zone (Zone 1001).

Step 3: Access AE and select 'Aquifer Elevations' tab.

Step 4: Go to 'Bottom Elevation/Thickness' area, select unit [m] and enter value [-10] for bottom elevation.

Step 5: Select 'Physical Properties' tab.

Step 6: In the 'Conductivity' field, change units to [m/day] and enter [5].

Step 7: Now select 'Zone 2001' under 'Layer 2' within the AE 'model explorer' pane.

Step 8: Select 'Aquifer Elevations' tab.

Step 9: Uncheck the 'Top Elevation' (see note at the end).

Step 10: Go to 'Bottom Elevation/Thickness' area, select unit [m] and enter value [-50] for bottom elevation.

Step 11: Select 'Physical Properties' tab.

Step 12: In the 'Conductivity' field, change units to [m/day] and enter [30].

Note: IGW will automatically assign the top elevation of bottom layer as the bottom elevation of top layer so you do not need to separately enter top elevation for bottom layer.

11.3. EDITING OBJECTS IN A LAYER

According to the conceptual model of Figure 1.2.2, the wells are screened in the bottom layer. The contamination plumes are also in the bottom layer. The following steps will show you how to copy objects from one layer to the other and how to delete/edit/add features in model layers.

Step 1: Use the Layer Selector sliding button to go to Layer 1.

Step 2: Click the 'Select a well and edit it' button and click on the Middle Well in the model area. The selected well turns red.

Step 3: Right click on the selected well. A menu pops up. Click on 'copy' in the popup menu.

Step 4: Use the Layer Selector sliding button to go to Layer 2.

Step 5: Right click any where in the model area of Layer 2. A menu pops up. Click on 'paste' in the popup menu.

Middle well is copied with all its attributes and location in Layer 2.

Step 6: Repeat steps 1 to 5 to copy East well into Layer 2 as well.

Step 7: Access AE.

Step 8: In the 'Model explore pane', right click on 'Wells 1001' in Layer 1. A popup menu appears. Click 'Delete' in the popup menu. The two wells under 'Wells 1001' are deleted from Layer 1.

Step 9: Within the 'Model explore pane' select Layer 2 and rename the wells as Middle Well and East Well respectively.

You have just copied the wells from Layer 1 to Layer 2 and deleted the same in Layer 1.

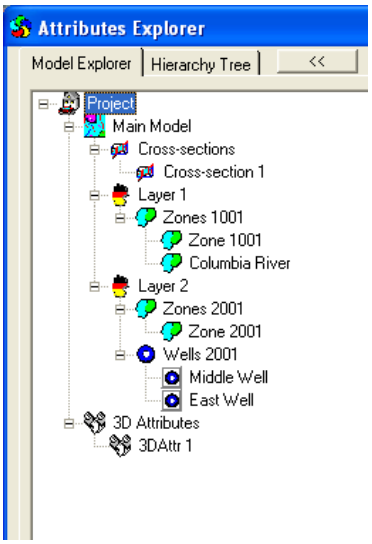
You cannot copy and paste particle from one layer to another. To remove particles from Layer 1:

Step 1: Use the Layer Selector sliding button to go to Layer 1.

Step 2: Click on 'Delete all particles' button to remove particles from Layer 1.

(You will add particles in Layer 2 at a later stage)

Up to this point, you have added an additional layer in your model, reassigned aquifer elevations and hydraulic properties to each layer, deleted particles and wells from Layer 1 and added wells in Layer 2. This setup conforms to your conceptual model of Figure 1.2.2. In the AE 'Model explorer pane' the model hierarchical structure now looks as shown in the opposite figure.



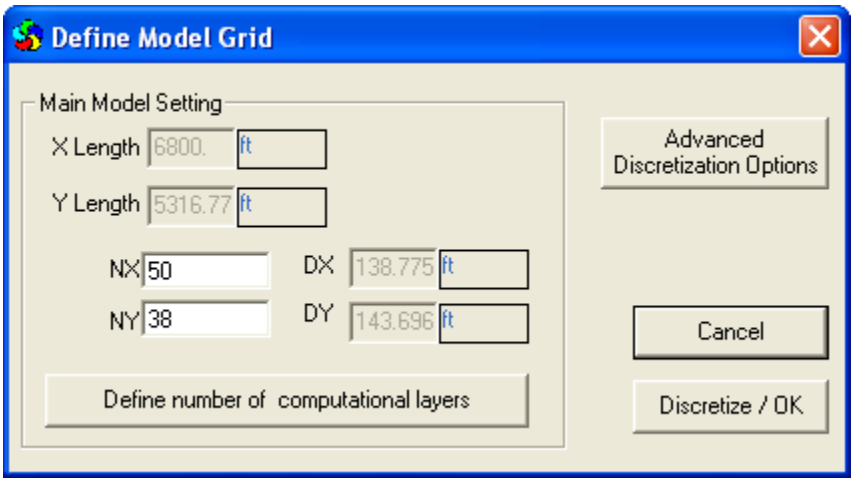
11.4. ADDING COMPUTATIONAL LAYERS

Computational layers help refine numerical simulations. Any model layer can be divided into desired number of computational layers. The following steps will show you how you can divide the top layer in 2 computational layers and bottom one in three.

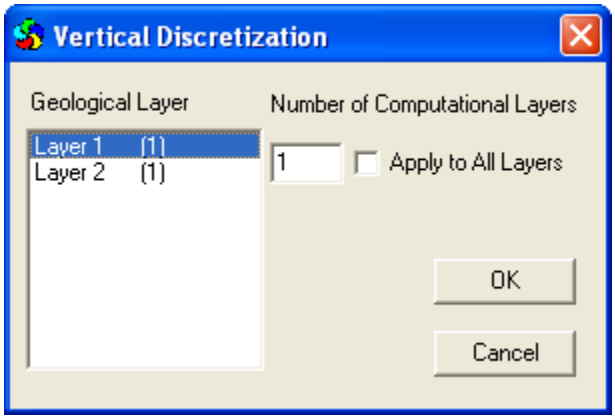
Step 1: Click the 'Set Simulation Grid' button

The ‘Define Model Grid’ window appears.

Step 2: Click the ‘Define Number of Computational Layers’ button.



The ‘Vertical Discretization’ window appears.



The layer data appears in the ‘Geological Layer’ area, where each conceptual layer is represented by an entry. The number of computational layers in that layer is indicated in parentheses. ‘ (1)’ indicates that there is one computational layer in ‘Layer 1’.

Step 3: While ‘Layer 1 (1)’ is selected in ‘Geological Layer’ area, enter the number [2] in the ‘Number of Computational Layers’ field.

Step 4: Select ‘Layer 2 (1)’ in the ‘Geological Layer’ area and enter number [3] in the ‘Number of Computational Layer’ field.

Notice the parenthetical numbers update as soon as you type in [2] or [3].

Step 5: Click ‘OK’.

You return to the ‘Grid Option’ window.

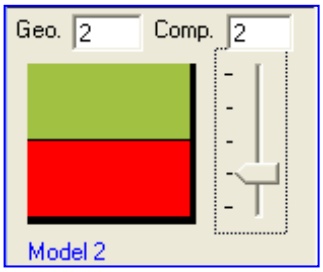
Step 6: Click ‘Discretize/OK’.

This will create 2 geological and 5 computational layers in your model.

11.5. PUTTING PARTICLES IN THE APPROPRIATE LAYER

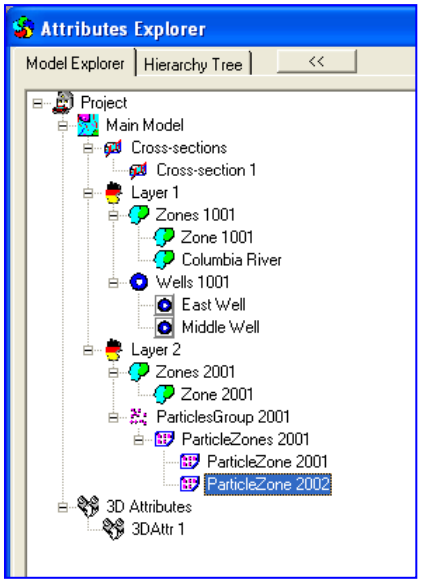
Recall that you had put the particles at Boeing and Cascade sites bottom third depth of the one layered model. In your multilayered model, you have to put the particle in the appropriate layer. In this case you will put the particles in the middle computational layer of the second geological layer.

Step 1: From the Layer Selector, move the sliding button to the middle computational layer of second geological layer



Step 2: Add particles at Boeing and Cascade sites as you did in section 9.1.

Notice the change in Model Explore plane of AE.



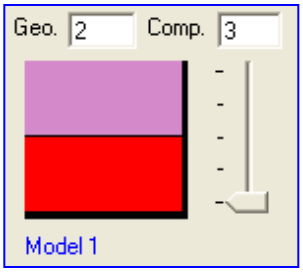
Step 3 Click the 'Forward' button

This will solve the model with added layers and particles placed in the second geological layer.

11.6. MODEL OUTPUTS WITH ADDED COMPLEXITY

We have just solved the model after adding one more model layer and dividing the two model layers in five more computational layers. IGW allows you to readily see the impact of this added complexity on the model solution for each computational layer. The software updates and redraws the working area and cross-sections as the model is solved.

(a) *Visualizing Head in Each Layer*:

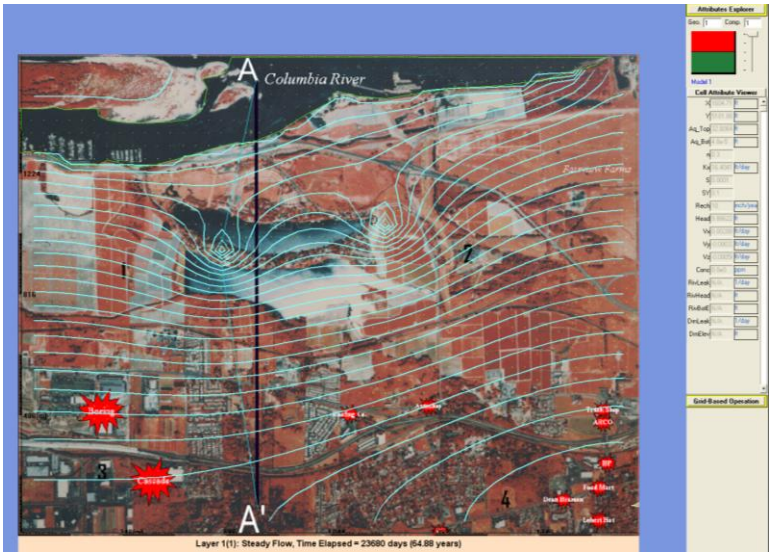


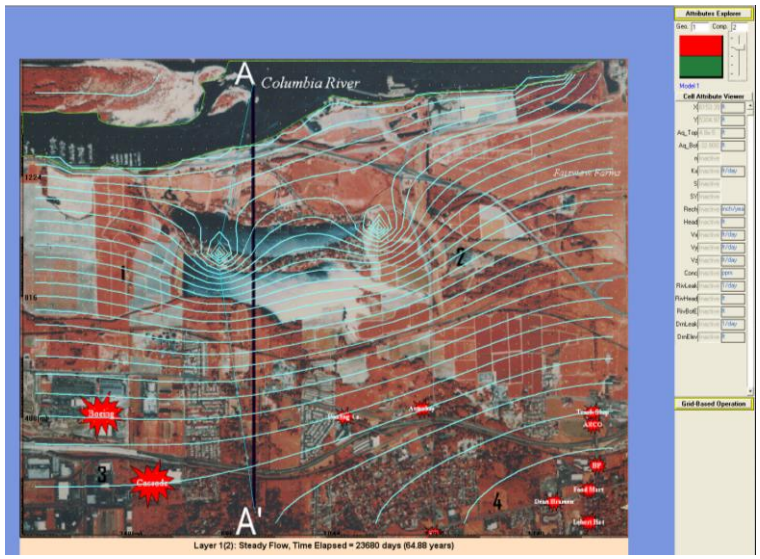
You can use 'Layer Selector' to move through the layers in model display area.

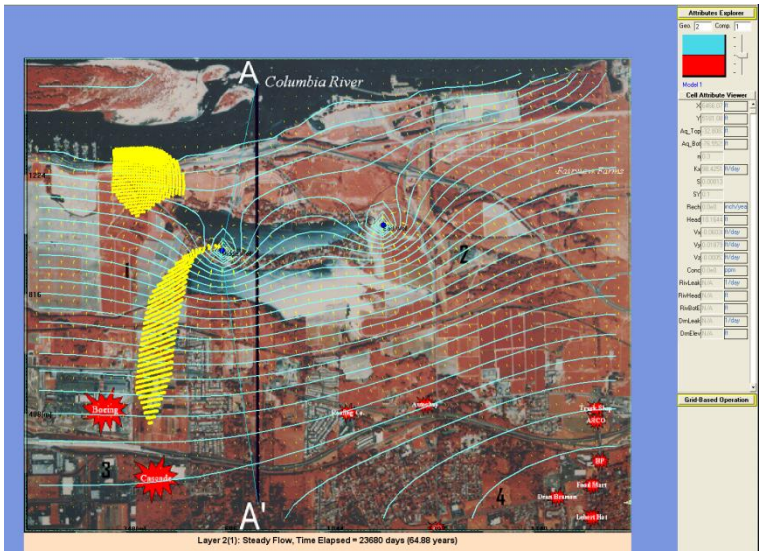
The Layer Selector is a simple tool for quickly accessing different layers within the model. As mentioned before, it is located in the upper right-hand corner of the IGW window. The 'Geo' and 'Comp' fields display, respectively, the geological and computational layer currently being accessed (and displayed in the Working Area).

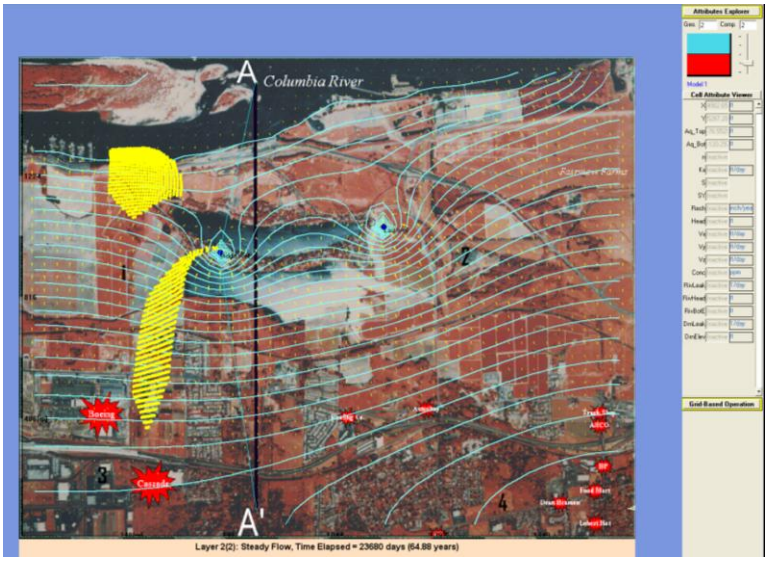
The color-filled square visually represents the geological layers with different (arbitrarily-assigned) colors. Intuitively, the top layer in the conceptual model corresponds to the top color (it can be thought of as a very crude cross-section). The arrow marker to the right of the square is a selector that can be clicked on and dragged to the desired layer (represented by a notch associated with each layer / color). The desired layer can also be selected by directly clicking on the notch

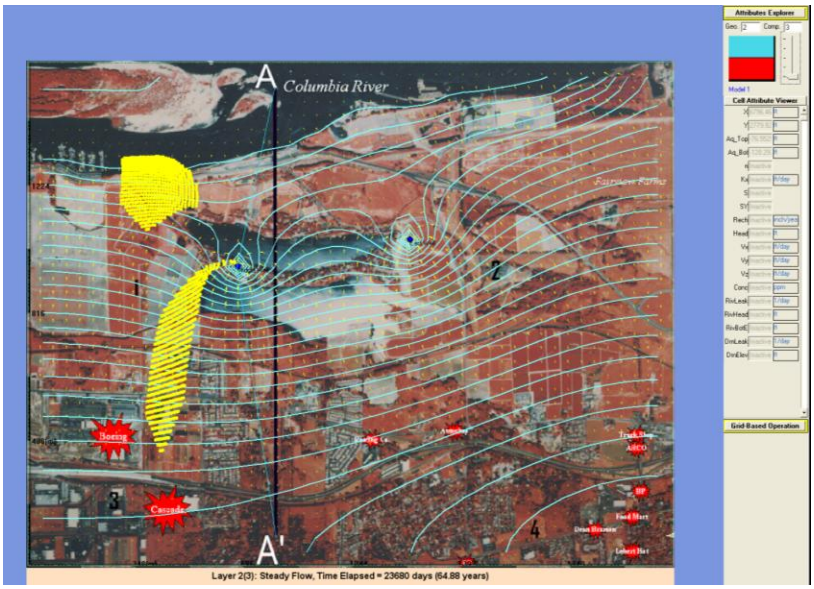
Notice that there are as many 'tick marks' along the layer selector sliding button as there are total number of computational layers [2 + 3 = 5]. You can slide the button to any 'tick mark' and the corresponding computational layer will be selected. The selected layer will display in the Working Area and will also be highlighted in the cross sectional view with a red border (see the cross section below). Notice the display in the top fields which show that 'Geo' layer '2' and 'Comp' layer '3' is selected. You can watch the plan view of the model as you move between the layers. For each computational layer, the model area presents its solution (head contours). You can observe how the solution differs in each layer. The solution for each computational layer is shown below:




A
Columbia River
A'
Boeing
Cascade
Layer 1(2): Steady Flow, Time Elapsed = 23680 days (64.88 years)


A
Columbia River
A'
Boeing
Cascade
Layer 2(1): Steady Flow, Time Elapsed = 23680 days (64.88 years)

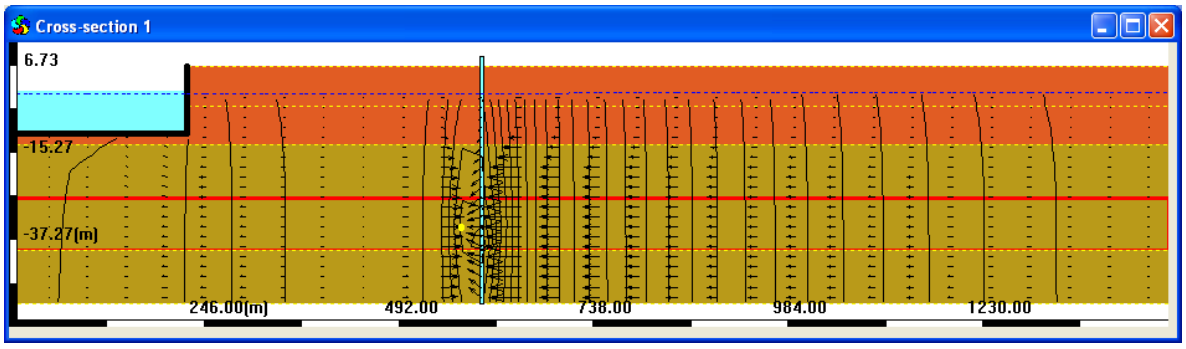

A
Columbia River
A'
Boeing
Cascade
Layer 2(2): Steady Flow, Time Elapsed = 23680 days (64.88 years)



From these display of model results, you will notice that by refining our model with more details, the model now predicts that the plume from Cascade site is captured by the Middle Well in approximately 65 years (recall that it was predicted 85 years by the simpler model).

(b) *Visualizing Cross Section*:

The computational layers are separated in the cross sectional view by dotted yellow lines. Notice the velocity arrows converging at the middle of the well, as opposed to the base of the well in the previous cross section. This refinement is achieved due to additional computational layers. IGW, by default, put the well screen in the middle-third of the layer in which it is located. The top and bottom computational layers in 'Layer 2' do not contain the well screen and you can see a more realistic picture of sub-surface velocity around the middle layer. As mentioned before, IGW calculates only one velocity for each computational layer. The varying velocity profile within the computational layers seen in the cross section is based on interpolation only.



At this point you can relate this cross section to the one-layer model cross section of Section 6.2, as well as the head contours for each computational layer to the one layer model of Section 6.1.

For any given computational layer there is only one velocity that is solved for. When you have more layers in the model (as you will add later on), the cross-section displays a continuum of velocities throughout the depth of a layer. Continuous velocity profiles are based on interpolation.

CHAPTER 12: USING SCATTER POINTS

IGW allows you to add information available at point locations within the model area, e.g., surface elevation, depth of geological layers, hydraulic conductivity etc. When information is available at more than one point in the model area, IGW can interpolate for the rest of the model area. The points on which information is available are called 'scatter points'. Scatter points allow you to add complexity to the model, as they are points where specific physical characteristics are known about the aquifer (or other features).

In the following sections, you will input aquifer elevation data at 4 locations (scatter points). IGW will process this input by interpolating elevations between these known points and updating the aquifer dimentions.

12.1. ADDING SCATTER POINTS

In the following example, you will use 4 scatter points in the model area (the point locations are marked as 1, 2, 3, and 4 on the basemap). At these points, you have the information for the surface elevation and the depth of overburden – Layer 1 in the model.

Point	Top (m)	Bottom (m)
1	7	-11
2	6	-12
3	20	-15
4	20	-18

The following steps take you through the process of adding scatter points.

Step 1: Use the Layer Selector to select the desired layer- in this example, choose Layer 1. (*Note*: When more than one computational layer is present, you may select within any desired conceptual layer and add features extending throughout all computational layers.)

Step 2: Click the 'Select a Zone' button. The cursor is now initialized to select a zone

Scatter points are always added in a zone. The zone in which you want to add scatter points must, therefore, be selected **before** adding the points.

Step 3: Click the mouse inside the parent zone (but not inside any other zone). The parent zone appears with red borders in the Working Area indicating that it is active.

Step 4: Click the 'Add Scatter Point' button.

The cursor is now initialized to add scatter points into the selected zone. (Note: even if you placed scatter points outside of the desired zone they would still be associated with the zone that was active at the time of adding the scatter points.)

Step 5: Click the mouse at the left side of each of the four locations, marked 1, 2, 3 and 4 on the basemap in the Working Area (after clicking, wait for the cross-hair cursor to reappear before clicking again).

The following figure shows the locations at which the points are added in the model. Note that the last point added next to '4' appears in bold red as it 'selected'. (To illustrated scatter points more clearly, the head contours and velocity vectors are turned off in the following figure by using 'Display Options')

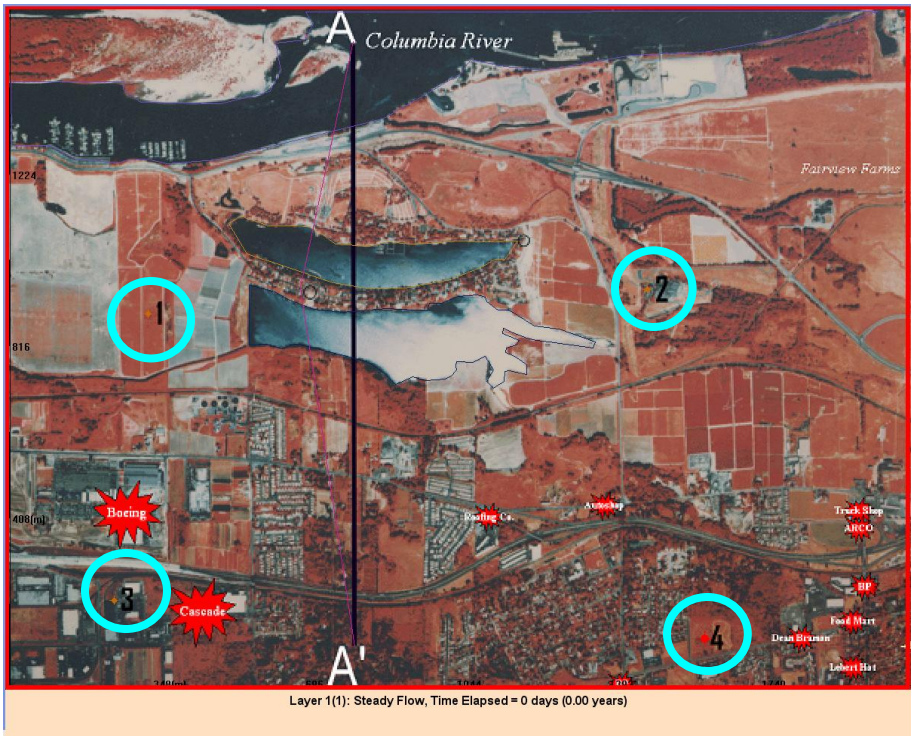
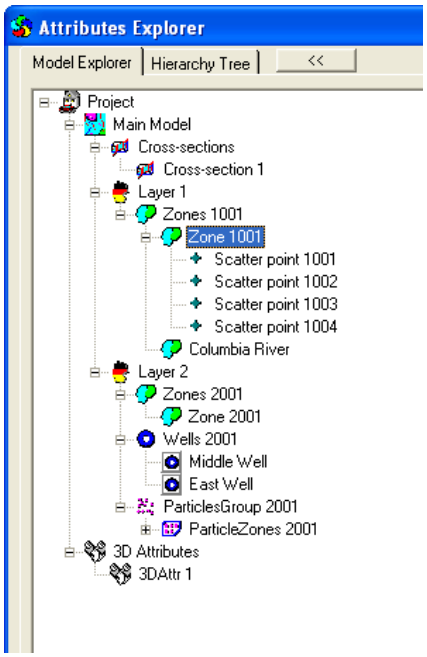


FIGURE 10.1.1 Scatter Point Data – Layer 1

Step 6: Access the AE.

You will see all the scatter points you added in Zone 1001 in the hierarchical pane of AE as shown below. (You can change the appearance of scatter point in the 'Scatter Point Style' area at the bottom left of the 'Attributes Pane' of AE).



Step 7: Select each scatter point in the 'Model Explore' pane and repeat the following:

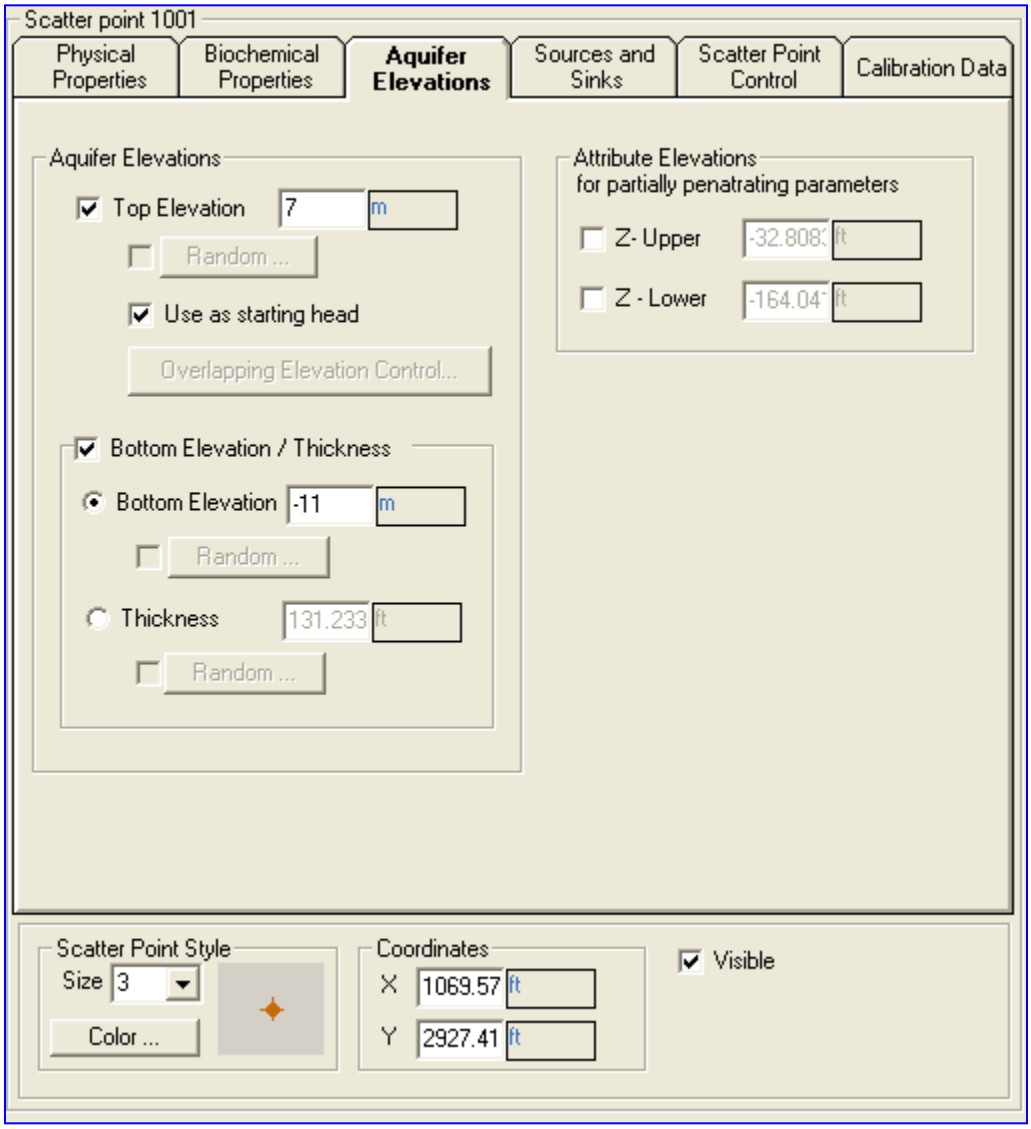
a) Select the 'Aquifer Elevations' tab.

b) Check the box next to 'Top Elevation'.

c) Select the unit [m] and enter the value of [7, 6, 20 or 20] (for Scatter Point 1001).

d) Check the box next to 'Bottom Elevation / Thickness'.

e) Select the unit [m] and enter the value of [-11, -12, -15 or -18].



Step 8: Select 'Zone 2001' in 'Layer 2' in the Model Explore Pane of AE.

Step 9: Select 'Aquifer Elevations' tab and un-check 'Top Elevation' box (if it not already unchecked).

(*Note*: This step is important because when top of the bottom layer is not defined, IGW assigns elevation of the bottom of top layer as the top of bottom layer. Now that the bottom of top layer (Layer 1) will be interpolated using scatter point information, the top of 'Layer 2' will not be a constant value and needs not be defined.)

Step10: Click the 'Set Simulation Grid' button to discretize the model. Define Model Grid window appears.

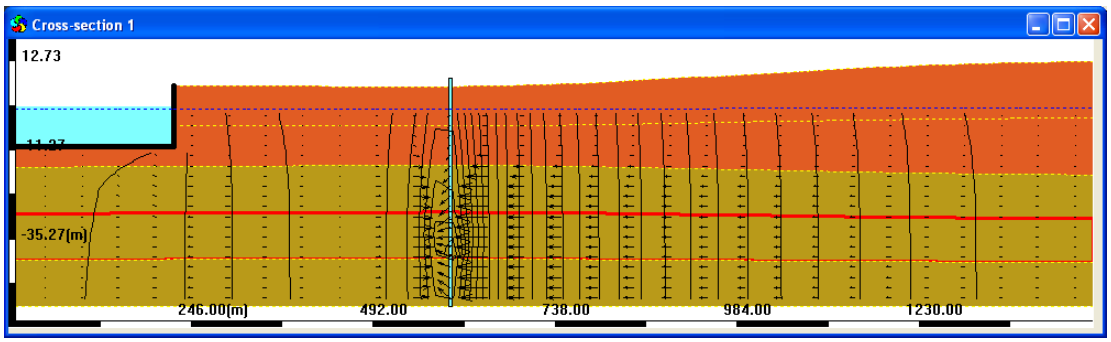
Step 11: Click 'Discretize/OK'.

Note: Whenever you make changes in a model, you can use 'Map into to Numerical Model' button and the changes will take effect. However, if there is any change in aquifer dimensions or grid size, this button will not fully implement the changes. You must use 'Set Simulation Grid' button.

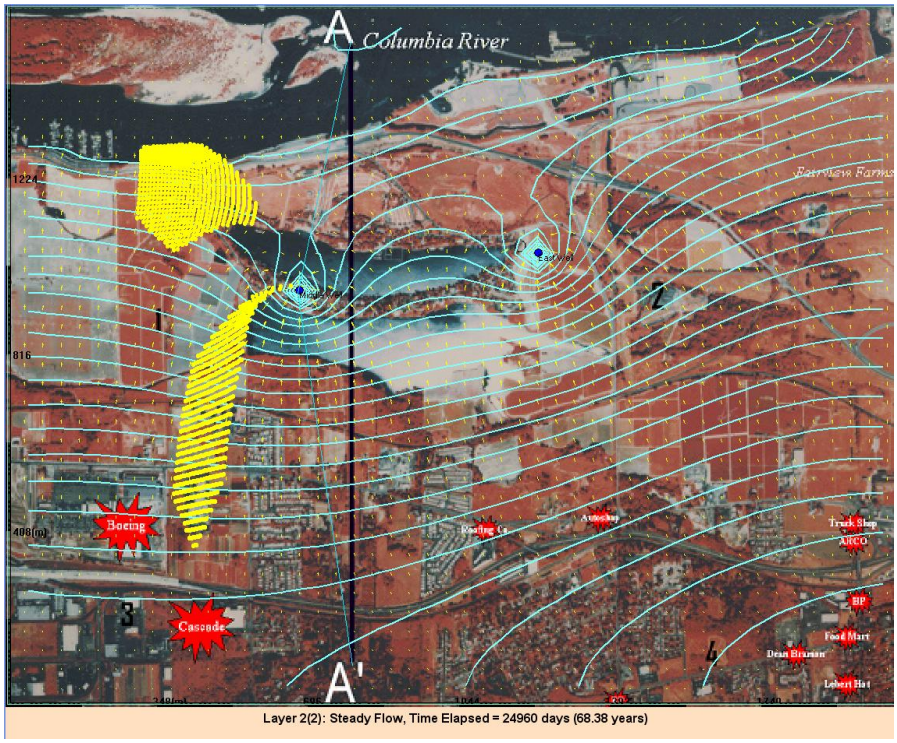
Step 12: Click the 'Forward' button to solve the model with the updated stratigraphy.

12.2. VISUALIZING RESULTS

Notice the change in cross sectional profile. The thickness of the layers is not uniform anymore.



You may notice further changes in the head contours, velocity vectors (and hence particle trajectories), and plume migration paths. Particle migration, head contours and velocity for the middle computational layer of second geological layer are given below. Notice that with improved aquifer stratigraphy, the model predicts that Cascade plume will be captured by the Middle Well in 68 years.



When using scatter points, the model reconciles the data using an option of different methods. The default setting is the Inverse Distance Weighted (IDW) method, and is the most commonly used. The IDW method is based on the assumption that the interpolating surface should be influenced the most by nearer points, and least by the farthest points.

CHAPTER 13: REFINING THE MODEL

Up until now, we have built is coarse model. Features like Blue Lake and Fairview Lake were not included in the model with an assumption that they are less significant. No dispersivity was applied to contaminant transport. Grid size was kept coarse.

In the following sections, you will include smaller modeling features like the lakes mentioned above. For contaminant transport modeling, you will add dispersivity to contaminant transport. Finally, you will refine the grid size and compare the model outputs with those of the coarser model.

You will be able to visualize the effects of refining at every step.

13.1. ADDING MORE FEATURES IN THE MODEL

You will add Blue Lake and Fairview Lake in the top layer of your model as conceptualized in Figure 1.2.2. To add these modeling features follow these steps:

Step 1: Use the Layer Selector to choose Layer 1.

Step 2: Click the 'Set display options' button.

Display options for Model 1 window apprears.

Step 3: In the 'Simulations Inputs and Results' area, uncheck 'Head' and 'Velocity' boxes.

This will clear head contours and velocity vectors and the basemap will become more clear.

Step 4: Click the 'Create a new arbitrary zone and assign property' button

The cursor is initialized to draw the new zone.

Step 5: Draw a polygon tracing along the edge of Blue Lake in the basemap as explained in Section 3.4

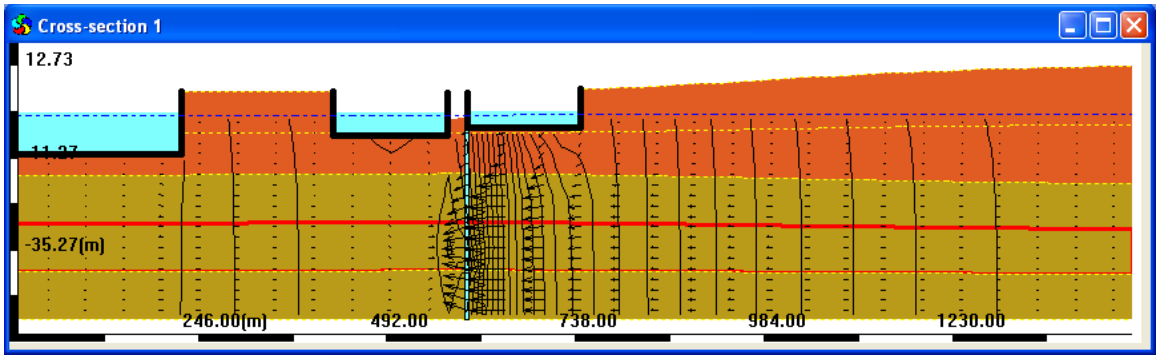
Step 6: Access AE and do the following:

(a) Click once on the newly created polygon in the 'Model Explore' pane and rename it 'Blue Lake'.
(b) Click on 'Sources and Sinks' tab, then select 'Head-Dependent Flux' tab.
(c) Check box next to 'Head-Dependent Flux (Two-way)'
(d) From the dropdown menu below the check box, select 'Lake'
(e) Go to 'Stage' field, select unit [m] and add value [3]
(f) Go to 'Bottom Elevation' area. Check radio button next to 'Constant', select unit [m] and add value [-2] in the field.
(g) Go to 'Leakance' area. Check radio button next to 'Constant', select unit [1/day] and add value [0.1] in the field

Step 8: Repeat steps 4 to 6 to add 'Fairview Lake' in the model. [Use 'Pond' from the drop down menu, stage as 3.5 m, bottom elevation as 0 m and leakance as 0.1 /day]

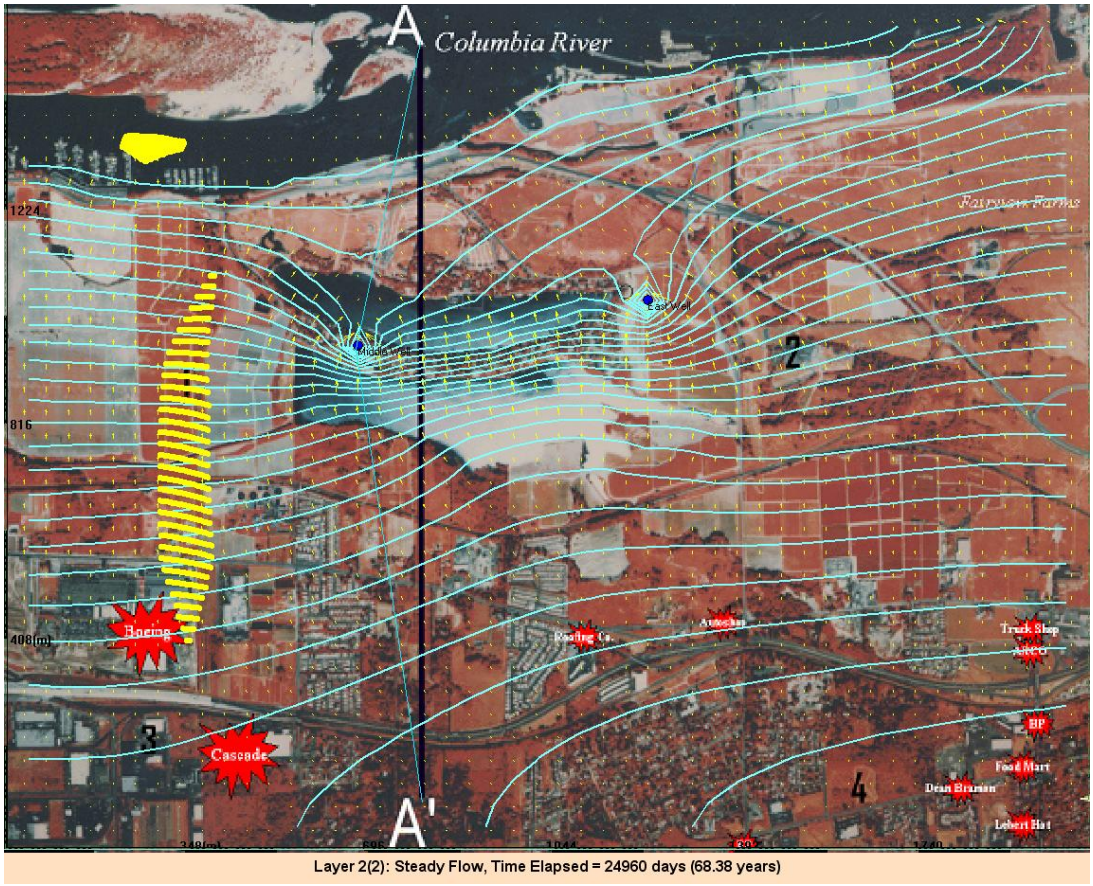
Step 9: Discretize and run the model.

The modeling results are shown below. Notice the change in cross section showing the newly added lakes with the Middle Well between the two.



You can see the impact on the shape of head contours in different layers using the layer selector. The following figure shows middle computational layer of the second geological layer. Notice that as the smaller lakes are added to the model, the path of contamination particles is changed. After 68 years of simulation, the Boeing plume ends up in Columbia River and Cascade plume, instead of being captured by the Middle Well, also seems heading towards the river.

Important: Note the position of the pumping well as it appears in the cross section. Although the well coordinates are exactly in the middle of the two lakes, yet it appears closer to the edge Fairview Lake. This is because the model shifts the well to the nearest grid-node. If the model grid is too coarse, the shift in the well location can be significant. You will be able to address this issue when you refine the model grid.



13.2. ADDING DISPERSIVITY

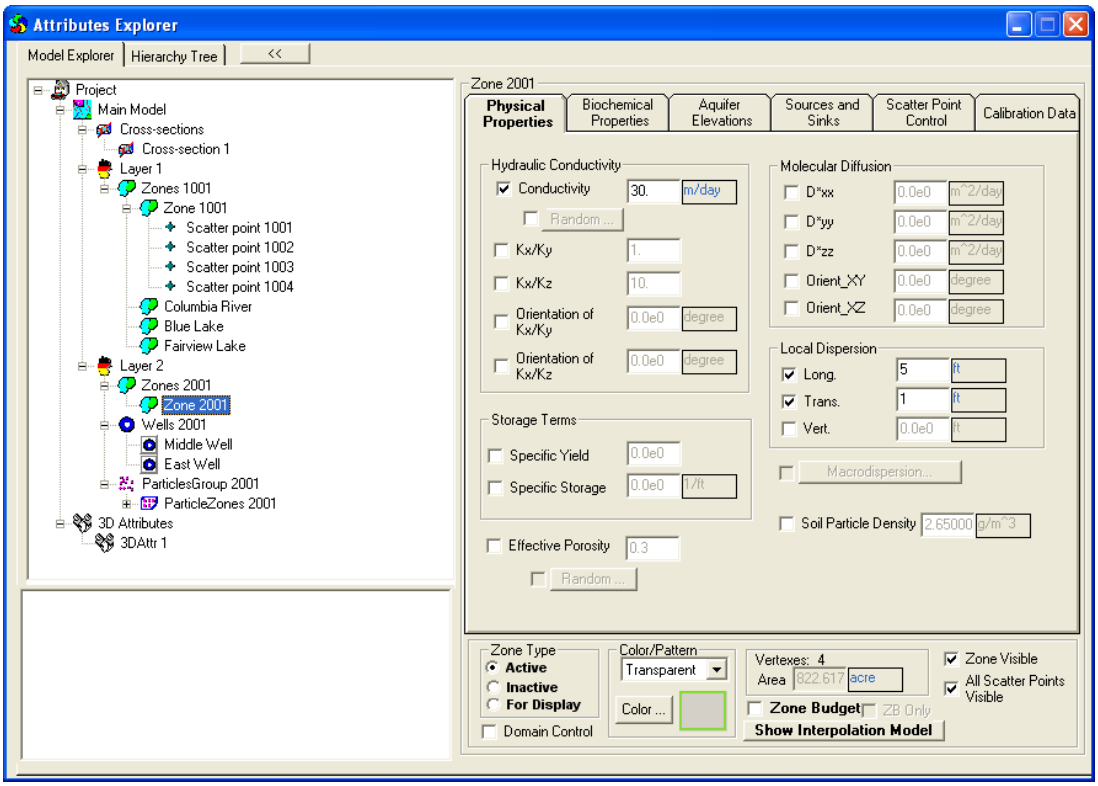
This section shows you how to perform particle tracking in the presence of dispersivity. The following steps walk you through adding dispersivity to the model and viewing the results in terms of the particle paths.

Step 1: Use the Layer Selector to choose the desired layer in which to add dispersion. For this example, select Layer 1 because this is the layer in which the particles are located.

Step 2: In the AE window, access the parent zone for that layer (in our example, this is Layer 2, Zone 2001).

Step 3: In the 'Local Dispersion' box on the 'Physical Properties' tab in the Attribute Input pane, select 'Long.' and 'Trans.' boxes.

Step 4: Adjust to the desired units [use the default] for the longitudinal (Long.) and transverse (Trans.) dispersivities and enter values in their respective fields [5 and 1, respectively].

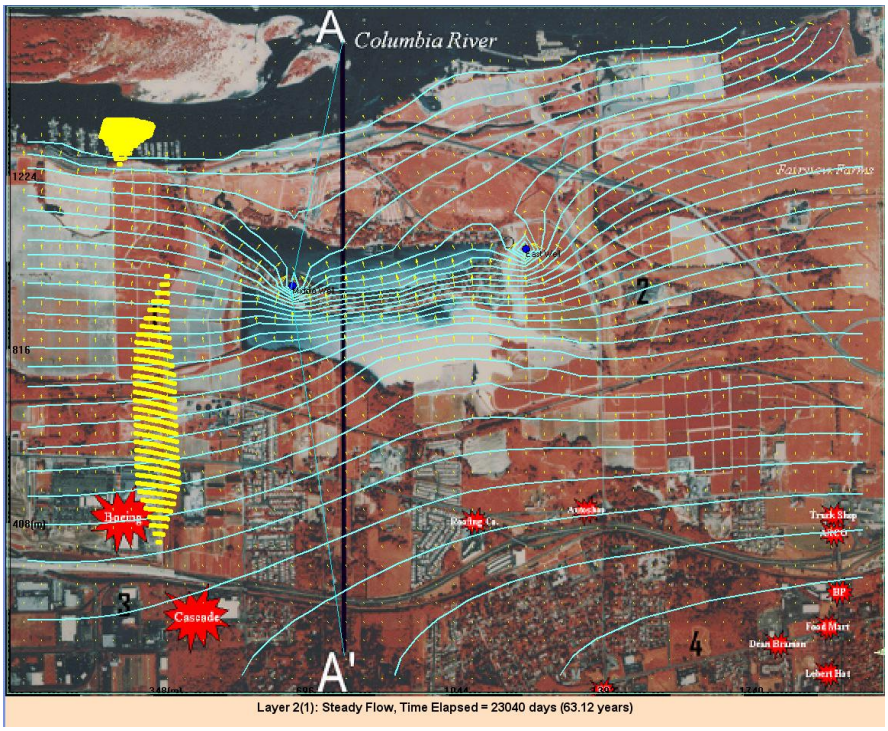


Step 5: Click the 'Map into a Numerical Model' button to discretize the model.

Step 6: Click the 'Forward' button to solve the model.

Notice any differences in the particle movement that changing the dispersivity values has caused.

[at this point IGW 4.7 and 5.0 are not implementing dispersion – the figure below does not show effects of dispersion]



Step 7: Stop the simulation by clicking the '[Pause](#)' button

Step 8: Click the '[Initializing Particles](#)' button to return the particles to their initial location.

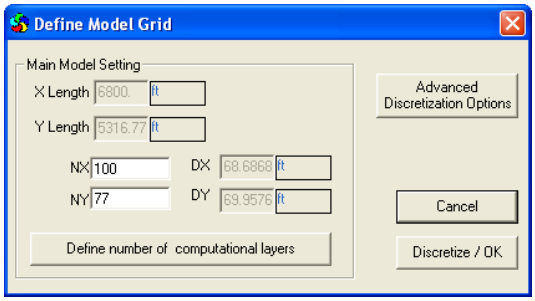
Step 9: Click the '[Reset Particle Clock](#)' button to reset the clock

13.3. REFINING GRID SIZE

You can change the grid size of the model at any time you want. By making the grid smaller, the model is more refined but it can increases the computational time/resources quite significantly – depending on the processor speed and memory.

The following steps will let you refine your grid size.

Step1: Click the '[Set Simulation Grid](#)' button to discretize the model. [Define Model Grid](#) window appears.



Step 2: In the 'NX' field, enter [100]. 'NY' field is automatically adjusted.

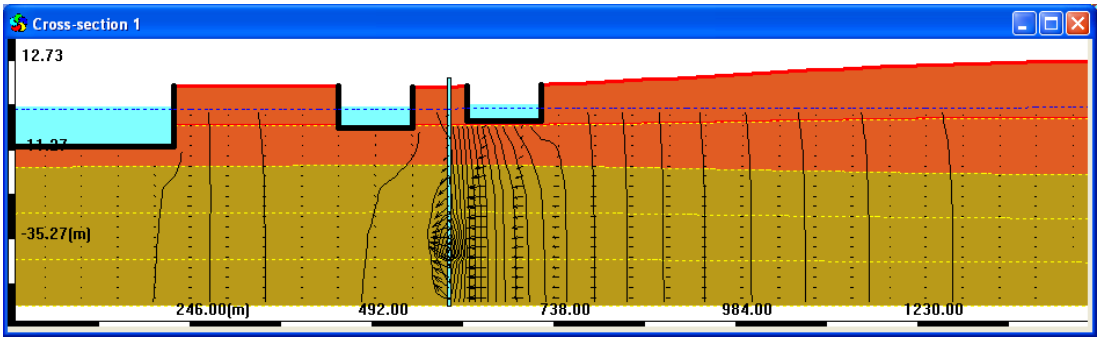
Step 3: Click 'Discretize/OK' button.

You will notice that now the model takes longer to discretize the model.

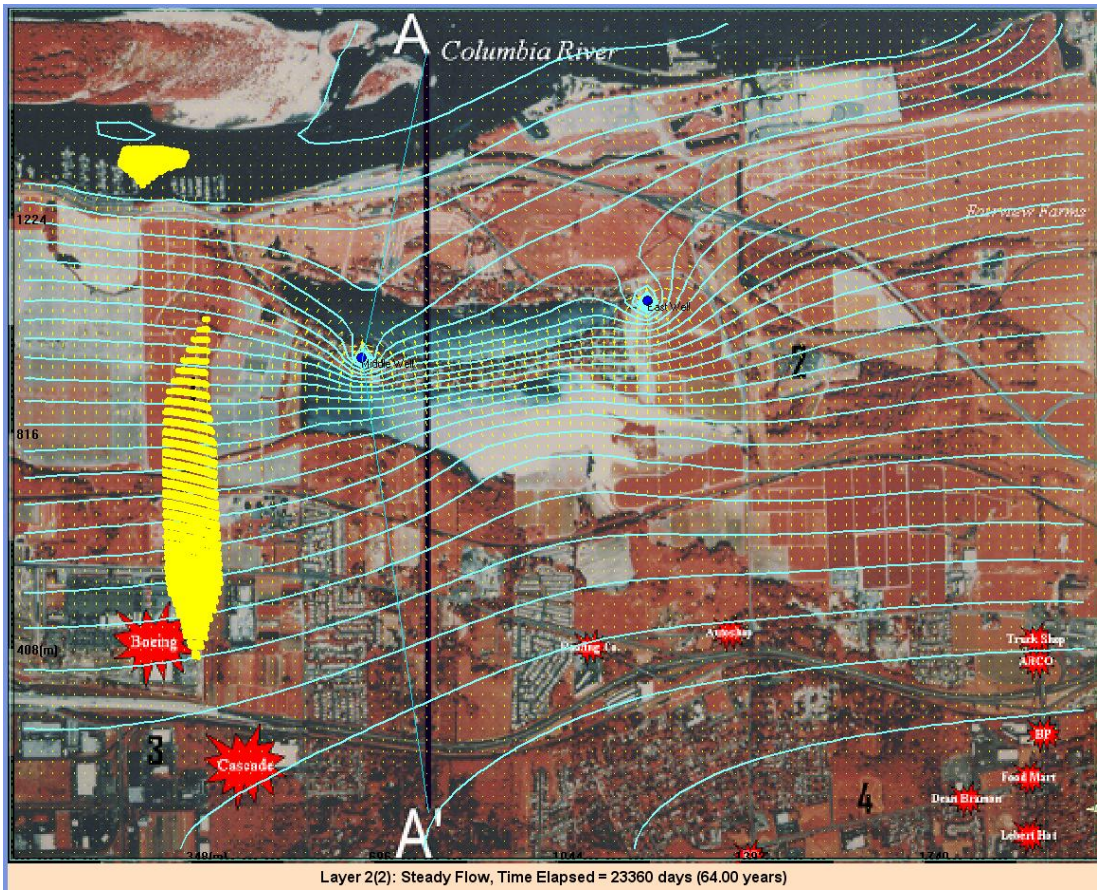
Step 6: Click the 'Forward' button to solve the model.

The model takes much longer to run each time-step.

Notice the change in cross section compared to Section 13.1. The well now appears in the middle of the two lakes. A finer grid can resolve the model features with more accuracy.



Particle movement, head and velocity for Layer 2 is shown below for the refined grid model.



CHAPTER 14: MODELING CONTAMINATION PLUMES

Instead of modeling contaminants as particles, we can also model them as a plume. This feature is especially useful if there is a continuous source of contamination.

The following sections will help you set up a contamination plume with a continuous source as shown in Figure 1.1.1. at the location of Autoshop.

14.1. MODELING A CONTINUOUS CONCENTRATION SOURCE

The following steps show you how to set up a continuous concentration source in the model.

Step 1: Use the Layer Selector to select second computational layer of Geological Layer 2, Assume the contamination represented by the particles has already passed upper layers.

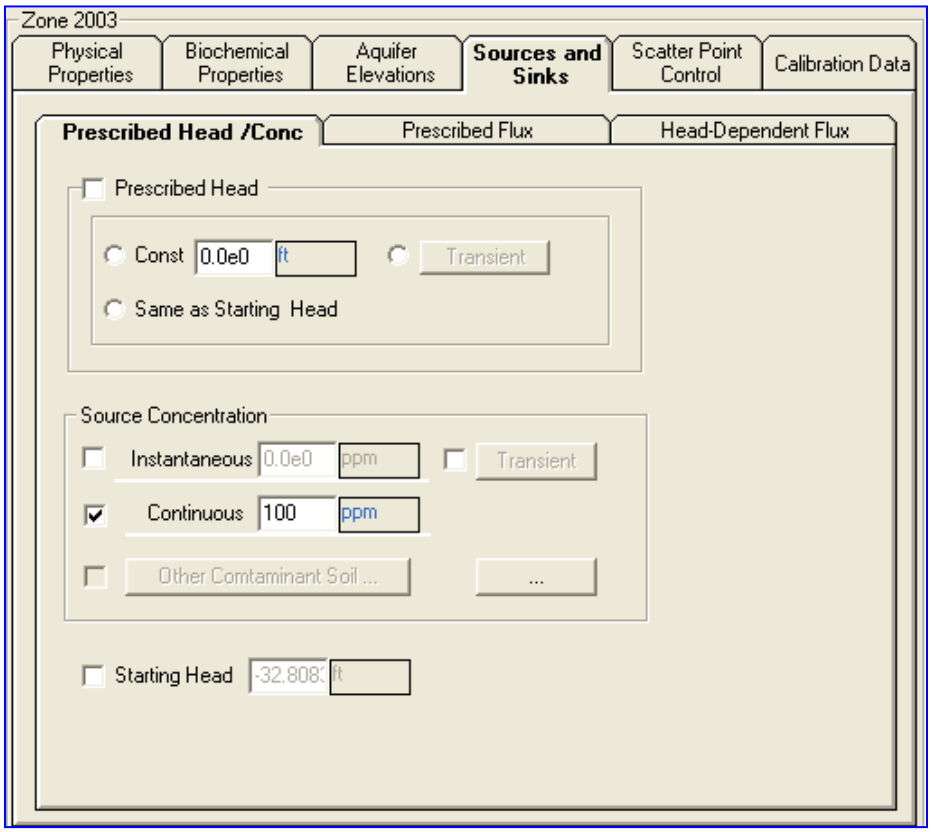
Step 2: Create a new zone where contamination might be located, using the 'Create new zone and assign property' button. Here, outline the area of the Autoshop on your basemap.

Step 3: Access the newly created zone in the AE window.

Step 4: Click on the 'Sources and Sinks' tab.

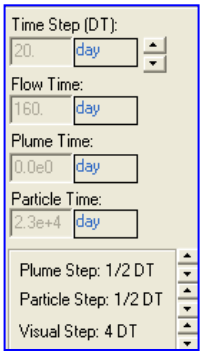
Step 5: In the 'Source Concentration' area, select the 'Continuous' box.

Step 6: Adjust the units field [ppm] and enter a value in the 'Continuous' field [100].



Solutions for concentration plumes are very dependent on the grid size and the time step. They may become oscillatory at a grid resolution that is too coarse, or with a time step that is too large. Now that we have a refined grid size, we can model the plume right away. You should increase the resolution of the grid and/or decrease the time step if your solution does in fact become unstable.

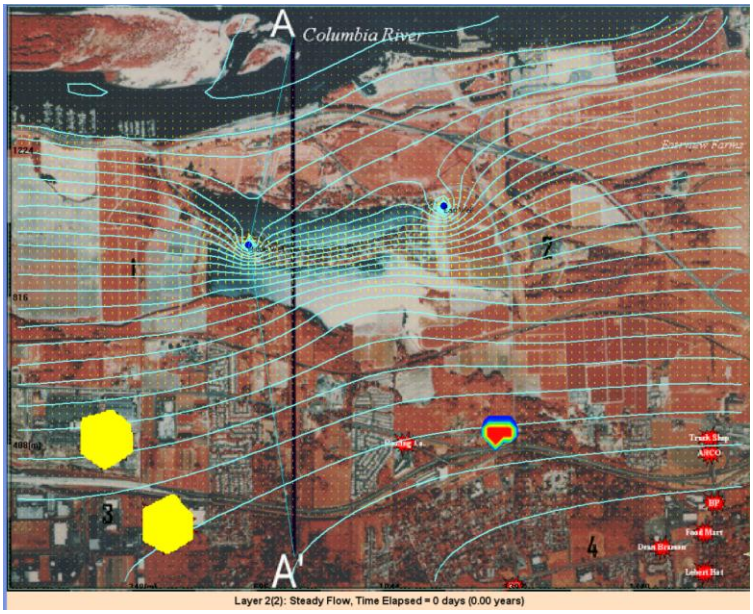
Step 7: In the time TPS area, make the time step equal to [20 days].



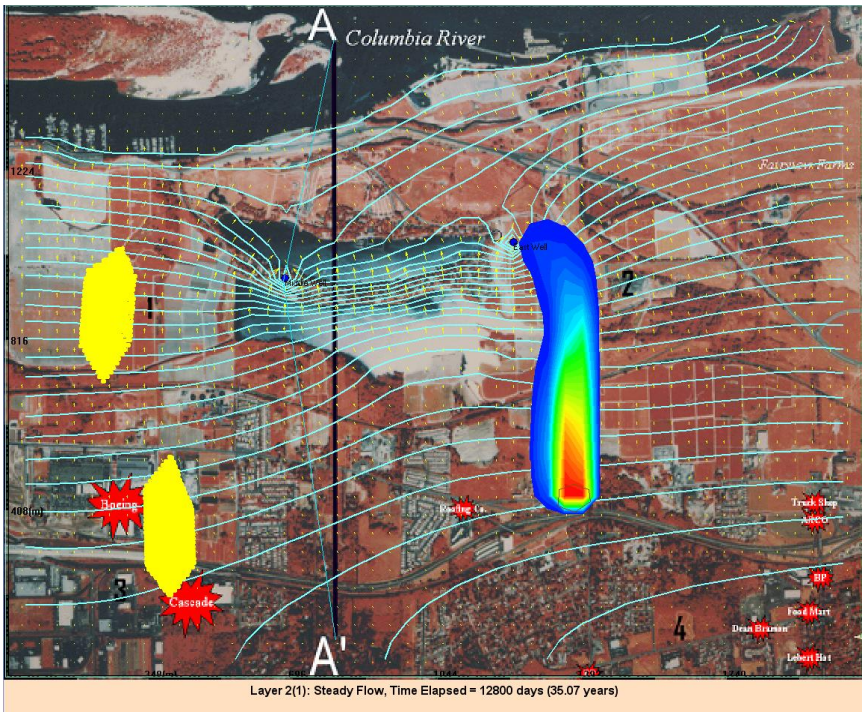
You may initialize the particle clock and reset the particles before proceeding to the next step

Step 8: Click the 'Map into a Numerical Model' button to discretize the model

Notice a plume appears inside the newly defined zone.



Step 9: Click the 'Forward' button to solve the model.
Notice how the plume spreads from the source



Notice that the plum is captured by East Well in approximately 35 years. Also notice that you can model particles and plumes simultaneously.

Note: there is a certain amount of 'numerical dispersion' in the software plume solution scheme. This causes the plume to disperse to a greater extent and faster than would be expected. Increasing the resolution of the model grid will reduce numerical dispersion. Note also that artificial dispersion was added in **Chapter 13** and further augments plume migration characteristics.

Step 10: Stop the simulation by clicking the 'Pause' button

14.2. INITIALIZING THE PLUME

Step 1: Click the 'Initializing Plume' button. (If at anytime this does not do anything, you will need to discretize the model.)

The plume returns to its original location.

14.3. INITIALIZING THE CONCENTRATION CLOCK

Step 1: Click the 'Reset Conc Clock' button

The time elapsed clock at the bottom of the main screen returns to 0 days unless there are other times being displayed (i.e. the flow time or particle time). In this case, you must also reset the clock with respect to the flow (needed only during transient simulations) [not applicable at this time] or the particles (see **Step 2** and **Step 3**).

Step 2: Click the 'Initializing Particles' button to return the particles to their initial location.

Step 3: Click the 'Reset Particle Clock' button

CHAPTER 15: UTILIZING A MONITORING WELL

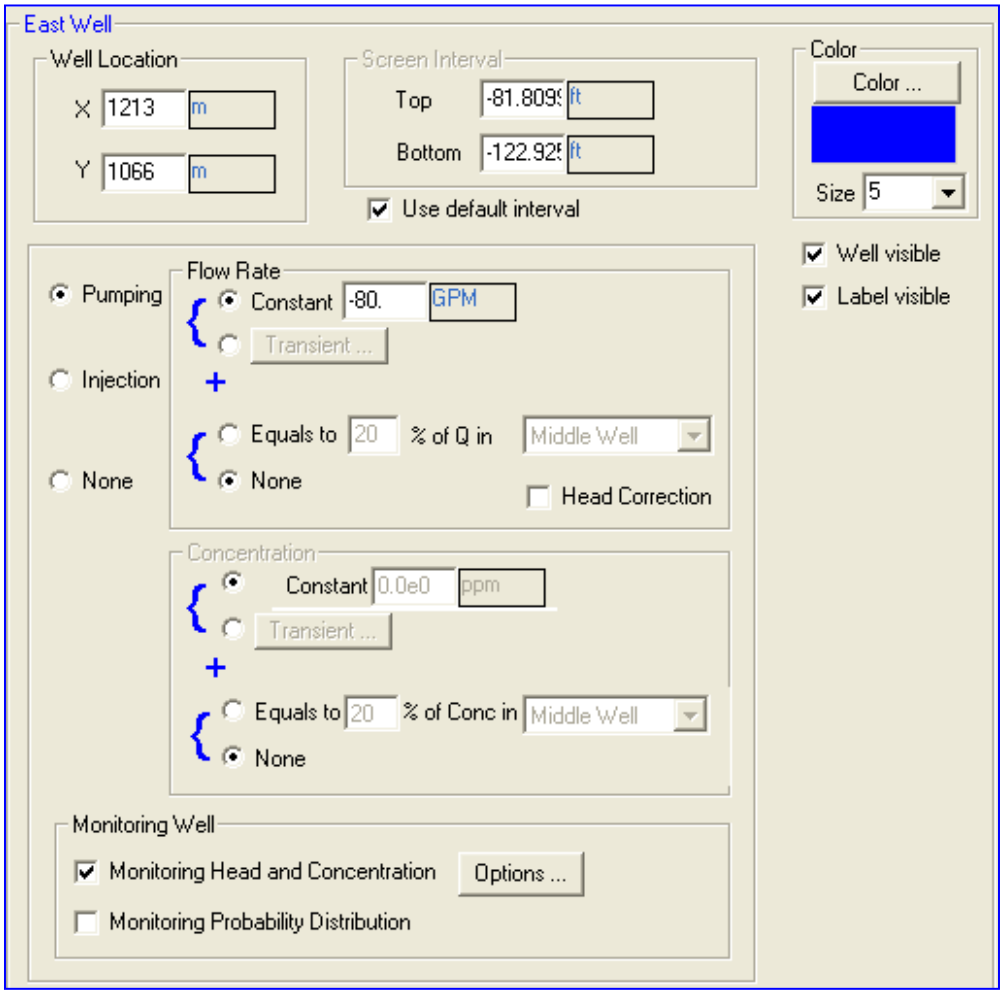
This chapter shows you how to setup and observe a monitoring well. You can use a monitoring well to observe both the concentrations and hydraulic head (in case of transient modeling). You can use a pumping well as a monitoring well and you can also create a well which is exclusively for monitoring heads and concentrations.

15.1. SETTING UP A MONITORING WELL

Monitoring wells are useful to examine the concentration of a contaminant at a point of interest. IGW 4.7 allows you to create monitoring wells that are purely for monitoring purposes or to add monitoring capabilities to wells that are pumping. The following steps show this procedure.

Step 1: In the Attribute Explorer window (AE), access the East Well that was created in Section 3.9

Step 2: In the 'Monitoring Well' area (in the 'Well Type' area), click 'Monitoring Head and Concentration'.



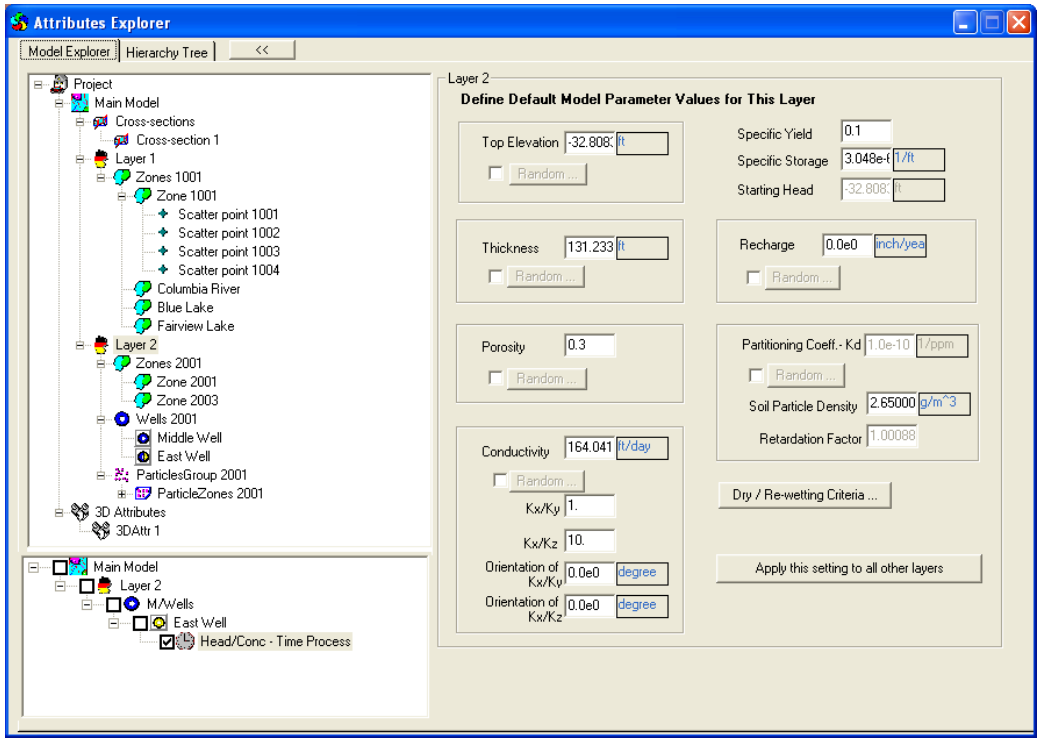
Step 3: Click the 'Convert the Model into a Numerical Model' button to discretize the model.

15.2. OBSERVING THE MONITORING WELL

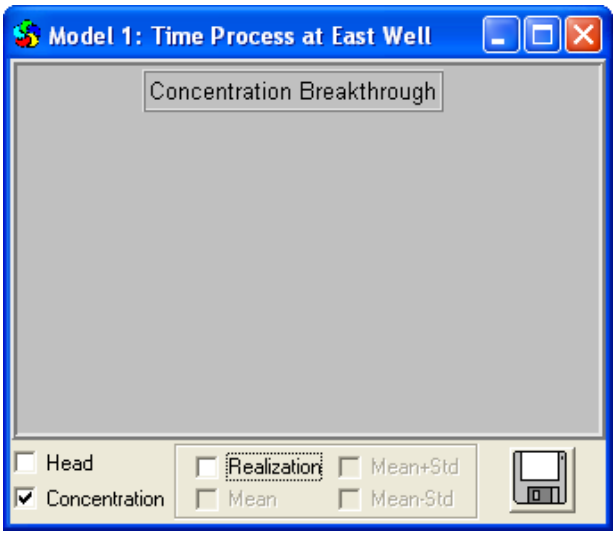
You could tell from the plume simulation viewed in **Chapter 11** whether or not the plume enters the monitoring well. However, it is more useful to observe the concentrations experienced in the observation well.

Note: You must have discretized the model to see the Time Process Selector in the next step.

Step 1: In the Time Process Selector (TPS), located in the lower left-hand corner of the AE, check box next to 'Head/Conc -Time Process'.



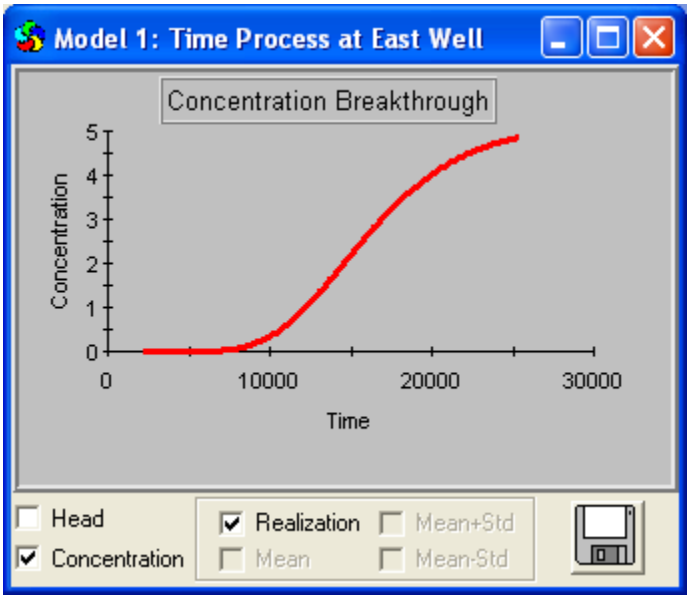
This opens up a window entitled 'Time Process at X'.



Step 2: Check box next to 'Concentration'.

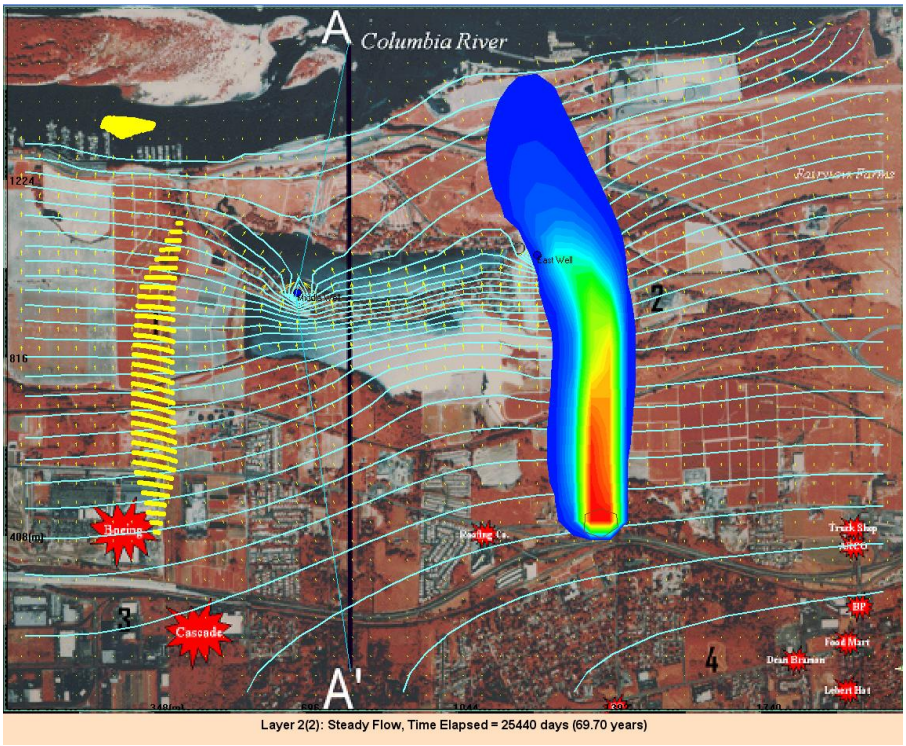
Step 3: Click the 'Forward' button to solve the model.

You can now view the concentration in the monitoring well as the simulation runs.



Note: the data displayed by the software is an average over the entire length of the well screen.

The above plot shows that the well starts to capture the plume after approximately 8000 days. The concentration in well is approximately 5 ppm after 25000 days. Recall that the source concentration is 100 ppm. The model predicts that most of the plume ends up in Columbia River.



Step 4: Stop the simulation by clicking the 'Stop' button.

The 'Time Process at X' window stays open but no longer updates You may close the window at any time after stopping the simulation.

Step 5: Click the 'Initializing Plume' button to initialize the plume.

Step 6: Click the 'Reset Conc Clock' button to reset the clock.

You must also separately initialize any particles (such as those added in **Chapter 8**) and reset the clock with respect to them.

Step 7: Click the 'Initializing Particles' button to return the particles to their initial location.

Step 8: Click the 'Reset Particle Clock' button.

CHAPTER 16: UTILIZING MASS BALANCES

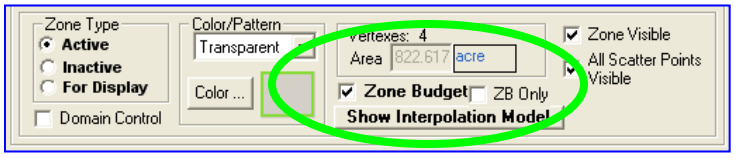
It is useful to understand the relationship between influxes and out fluxes for water or contaminants for a given area of interest. IGW allows you to examine/evaluate mass balance for the entire model are or any part of the model defined by a polygon. You can simultaneously select a number of areas within your model for mass balance. The areas may or may not be overlapping.

16.1. SETTING UP AN AREA FOR MASS BALANCE

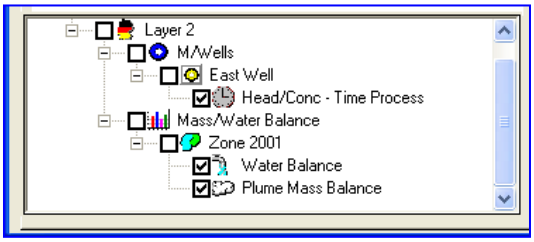
It is revealing to examine the mass balance for the entire active model area, to get an idea of the "big picture" of water input and output levels. You can draw a new polygon or select an existing one for mass balance. The following steps show you how you can utilize the existing polygon which defines the entire model domain. Water budget will be evaluated only for the layer in which the polygon exists.

Step 1: In the Attribute Explorer window (AE), access the second layer and select "Zones 2001".

Step 2: In the bottom middle of the AE, check the box next to "Zone Budget."

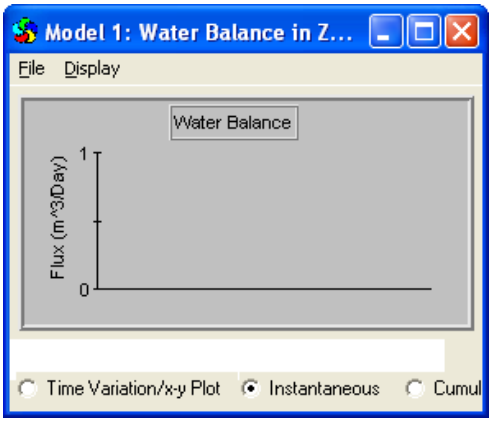


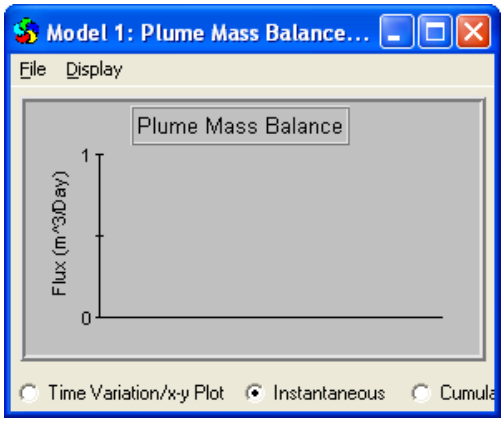
Step 3: The Mass Balance window appears in the lower left hand pane of the AE.



Step 4: Discretize the model.

Step 5: Check the boxes next to 'Water Balance' and 'Plume Mass Balance'. This opens two new blank windows that show the Water Balance and the Plume Mass Balance in Zone 2001.





16.2. OBSERVING THE MASS BALANCE

Step 1: Click the 'Forward' button to solve the model.

Watch the graphical representation of the mass balances. In the 'Water Balance' window, notice the flux associated with 'CHead'. This is the amount of water contributed from any defined Prescribed Head features in the layer [Columbia River]. Notice also that the software indicates the quantities associated with flux to and from layers above and below.

Step 2: Click the 'Stop' button to stop the simulation.
You may close the windows at any time after stopping the simulation.

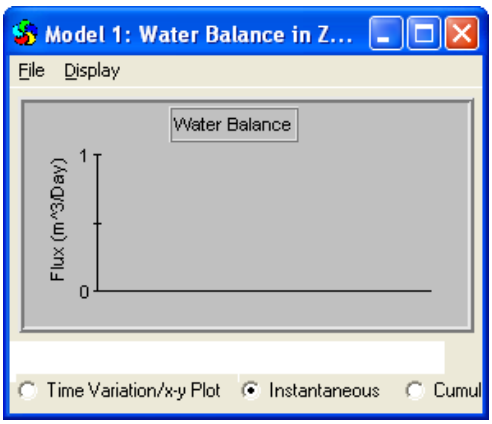
Step 3: Click the 'Initializing Plume' button to initialize the plume

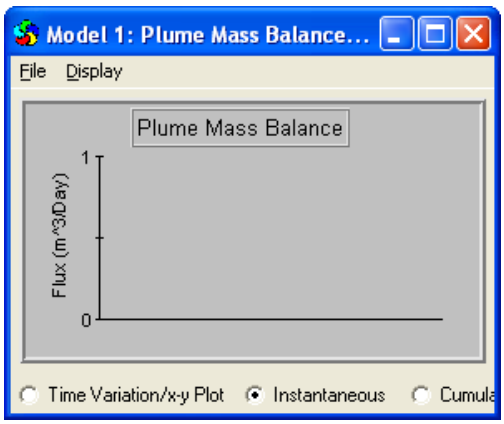
Step 4: Click the 'Reset Conc Clock' button to reset the clock.

You must also separately initialize any particles (such as those added in **Chapter 9**) and reset the clock with respect to them.

Step 5: Click the 'Initializing Particles' button to return the particles to their initial location.

Step 6: Click the 'Reset Particle Clock' button.





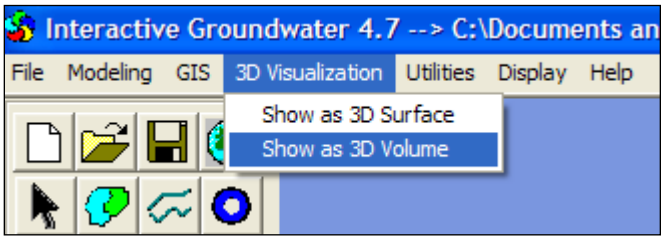
CHAPTER 17: VIEWING THE MODEL IN THREE DIMENSIONS

The software allows the user to view results in three dimensions, and manipulate the view by rotating and cropping/transposing the image as desired. 3D Visualization in IGW is dynamic: every time it is executed, the model is updated immediately with results displayed on the spot. This example gives a very brief overview of these capabilities.

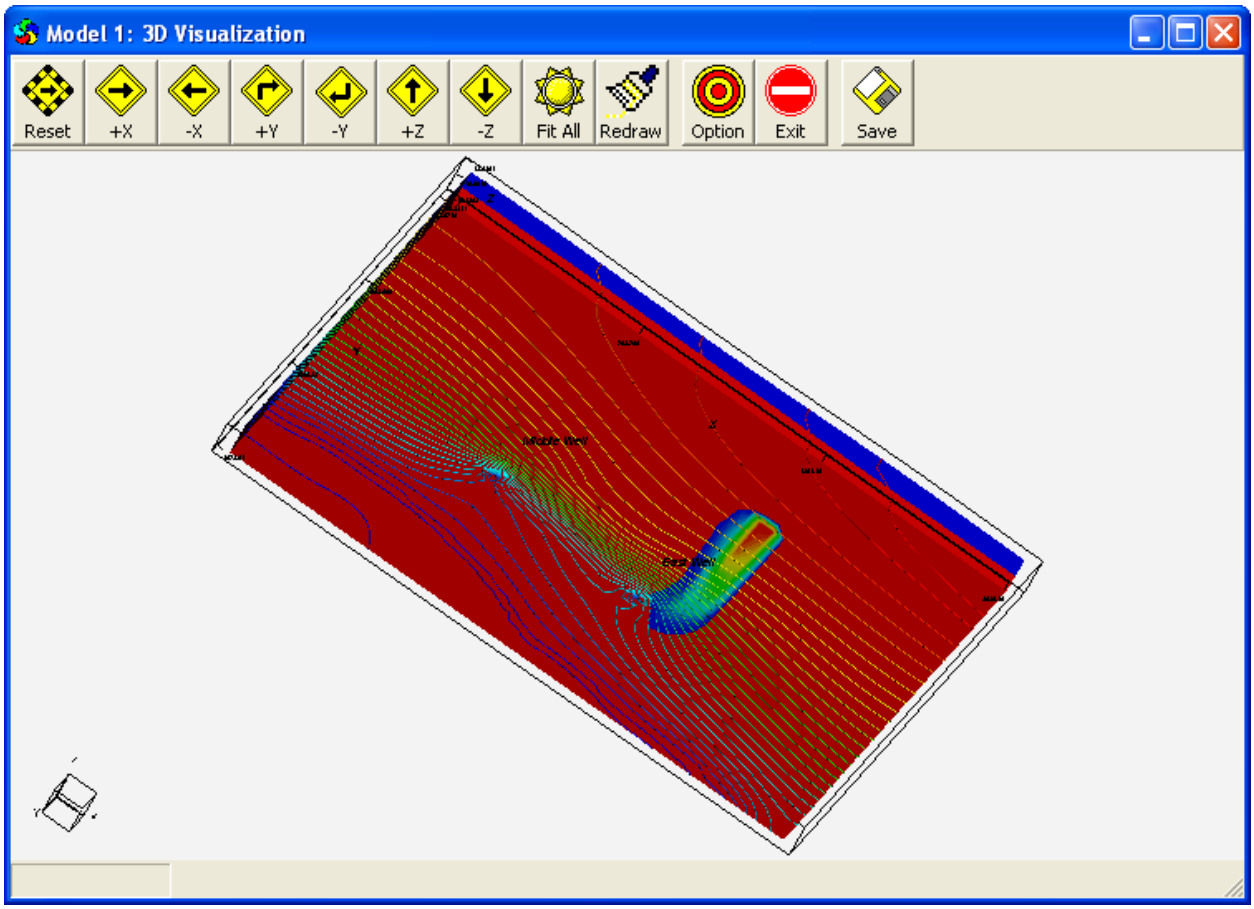
17.1. INTRODUCTION AND BASIC FEATURES

Following steps will guide you through 3D visualization process:

Step 1: Click on the "3D Visualization" tab at the top of the screen, and then click "Show as 3D Volume".



A new screen will appear. (Note: this screen may appear behind your current view, in which case minimize the working area to reveal this window.)



a.) In the window, left-click anywhere on the outside of the image to turn it any direction in three dimensions.
b.) Right-click in the inside to the right of the image to zoom in, and right-click on the left inside of the image to zoom out.

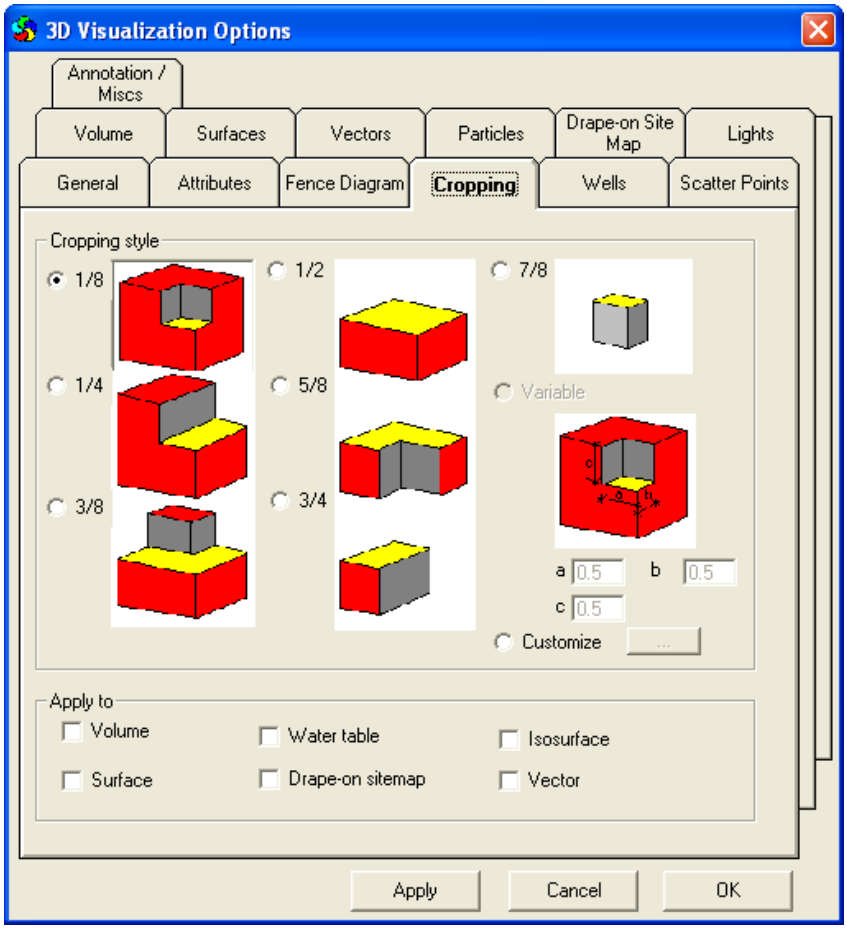
Also note that you can clearly view both layers of the model, as shown by different colors.

Step 2: Click the "Option" button on the top row, revealing a window with many different ways of manipulating the model.

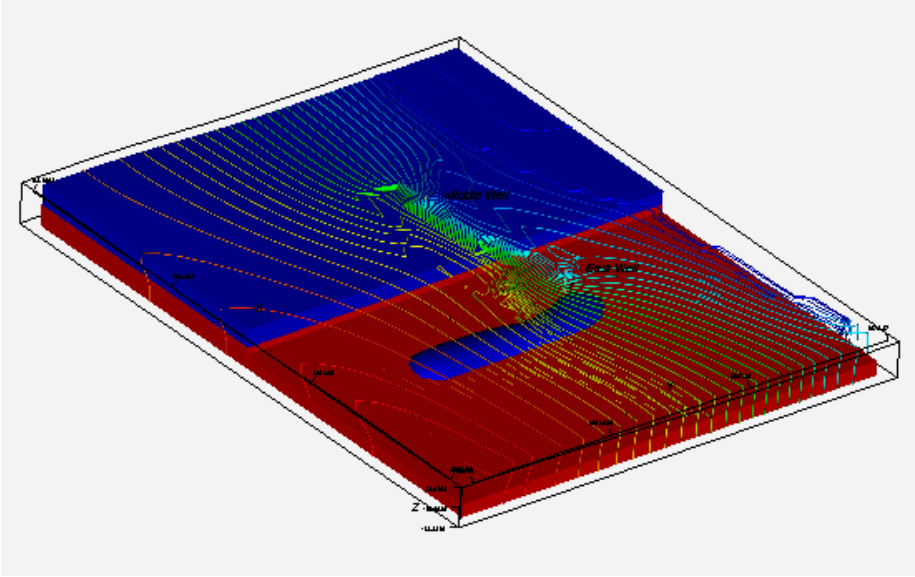
For this example, we will look at the Cropping and Fence Diagram functions and their associated graphics.

17.2. THE CROPPING FUNCTION

Step 1: Select the "Cropping" tab from the above-mentioned "3D Visualization Options" menu. You will see many different ways that IGW Version 4.7 can dissect the model.



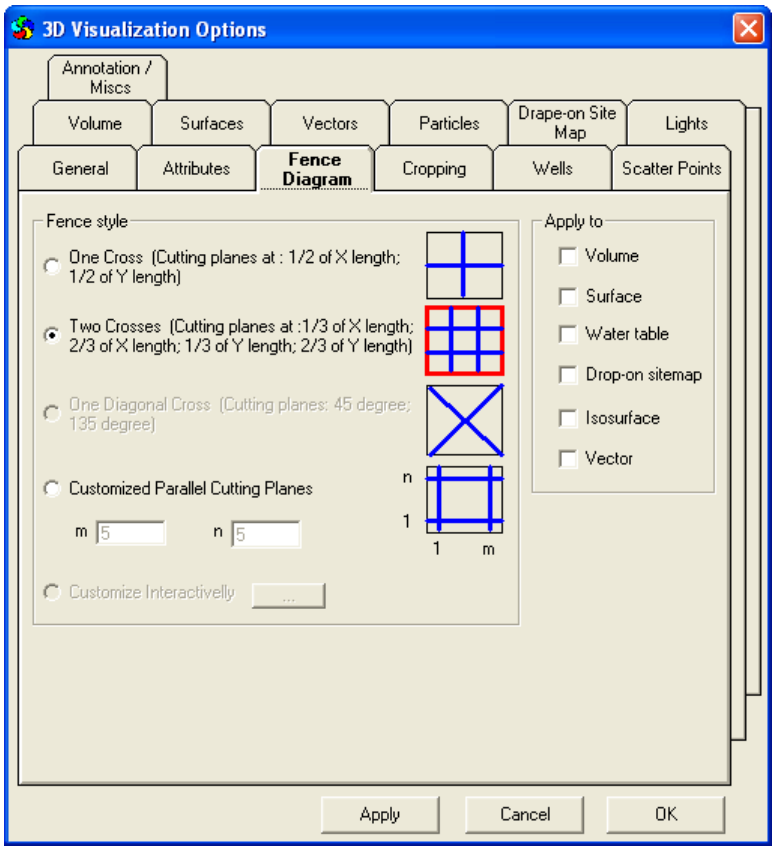
Step 2: For this example, select the "1/4" button, and then check "Volume" in the "Apply To" menu located directly below the cropping styles. Click "Apply", then click "Ok".



Step 3: You can manipulate the model as described in earlier steps of this section, to zoom in and out, and rotate the image to the desired position. Notice that the right ¼ of the top layer of the model has been removed, as requested.

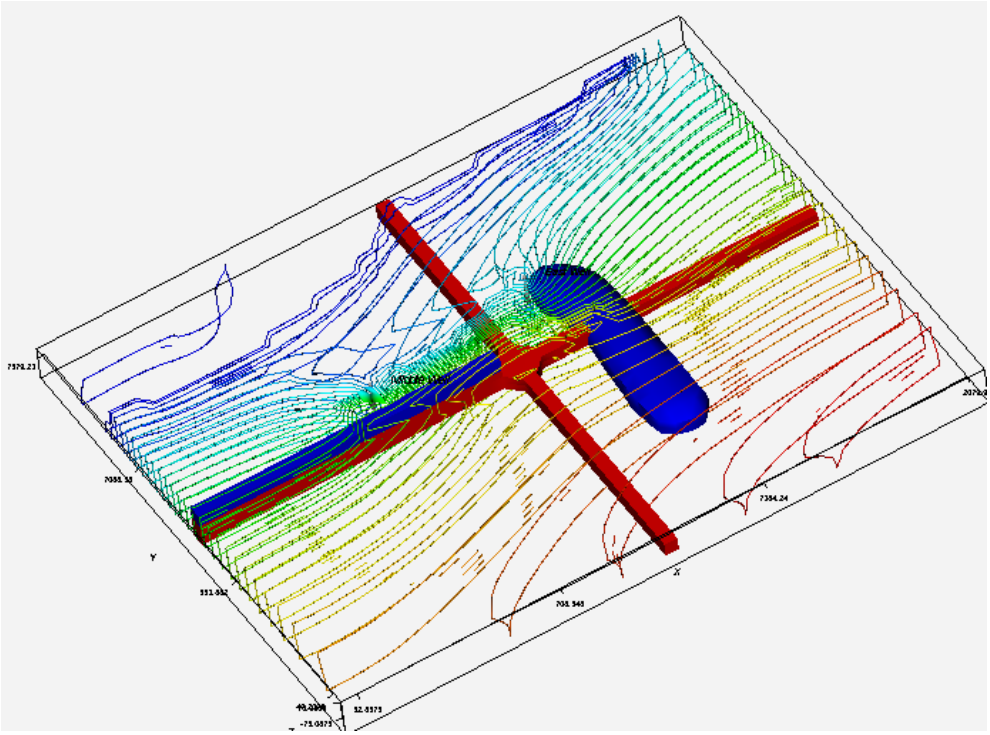
17.3. FENCE DIAGRAMS

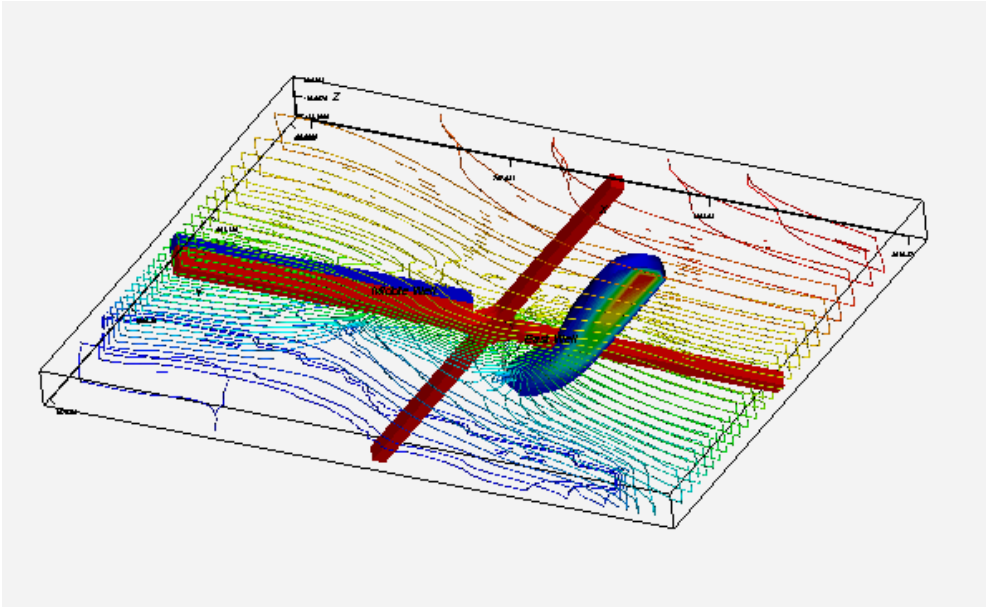
Step 1: Go back into the "Option" menu, and select "Fence Diagram".



Step 2: Under “Fence Style”, select “One Cross” and then in the adjacent menu, check ‘Volume’ in the “Apply to” region. Click “Apply” then click “Ok”.

Two views of 3D fence diagram are shown below, top one is looking the model from the top and the bottom one is looking the model from below. Notice that the constant concentration plume applied in the bottom layer is better visualized in the bottom view.





Step 3: When satisfied with the result, close the 3D Visualization window.

CHAPTER 18: SIMULATING TRANSIENT CONDITIONS

Up to now we have been simulating steady state flow. This chapter shows you how to run a simulation in transient state.

18.1. CHANGING SIMULATION TIME PARAMETERS AND SOLVING

This section will show you how to switch the model from steady state to transient state.

You need to add a transient stress to the model. (The following steps show you how to add transient features to a pre-existing Prescribed Head body of water. Of course, there are a myriad number of other transient stresses that can exist and more information about adding such features can be found in the IGW User's Manual.)

Step 1: Use the Layer Selector to select the desired layer. For this example, select Layer 1.

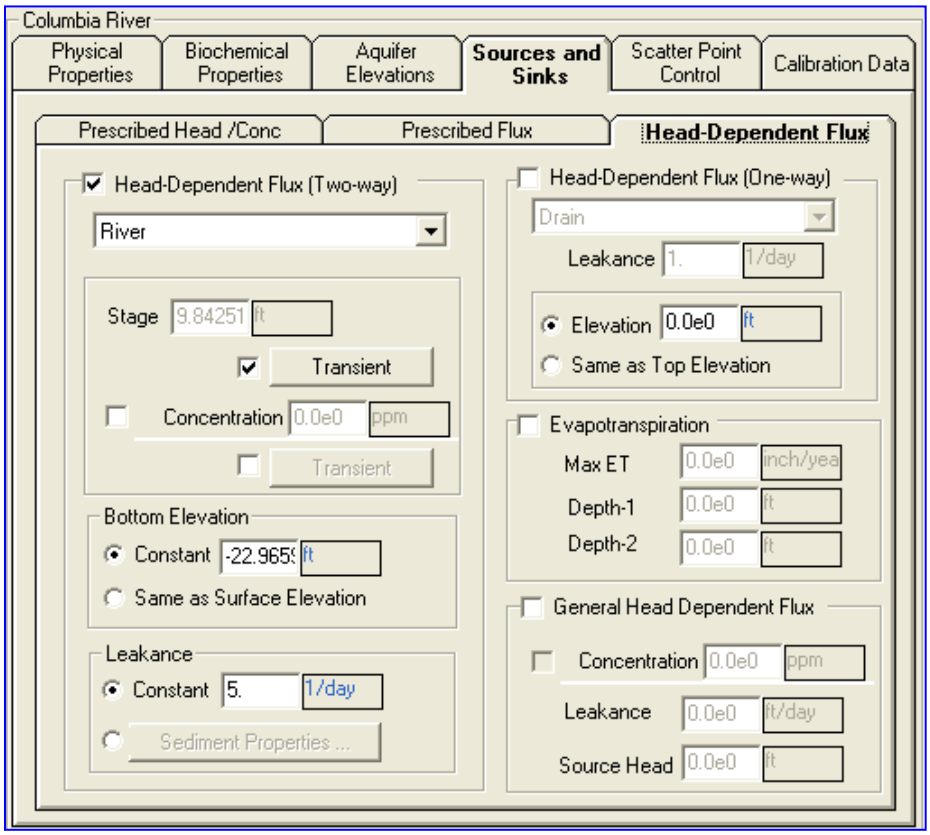
Step 2: Access the Attribute Explorer window (AE).

Step 3: Select a zone that corresponds to a head-dependent body of water, this being the Columbia River here in the example.

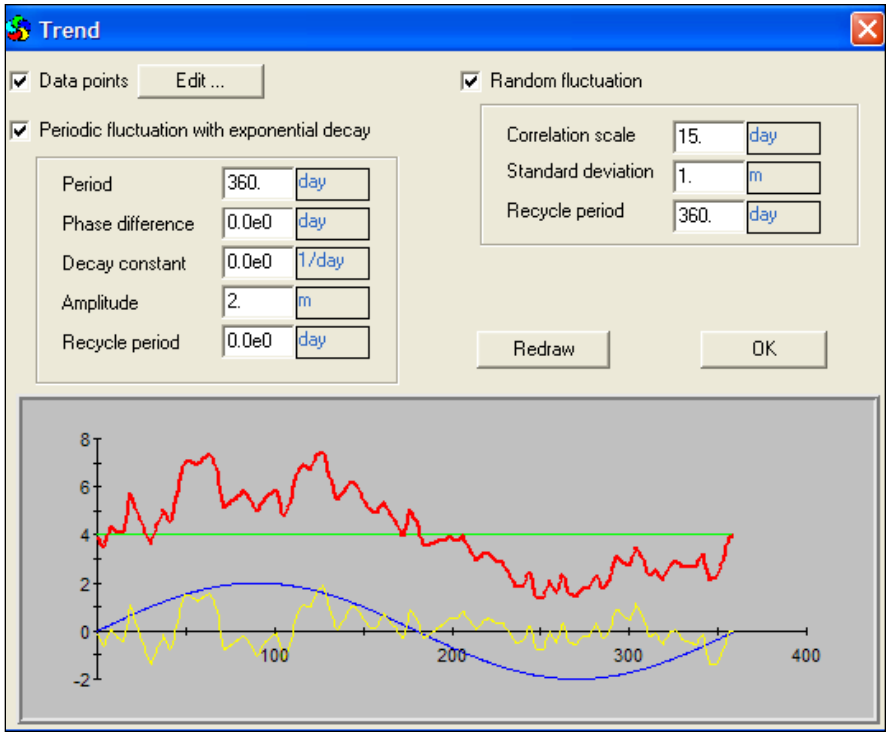
Step 4: Click on the 'Sources and Sinks' tab in the Attribute Input pane to access that level.

Step 5: In the 'Prescribed Head' area, select the disabled 'Transient…' button.

Step 6: Click the newly enabled 'Transient…' button.

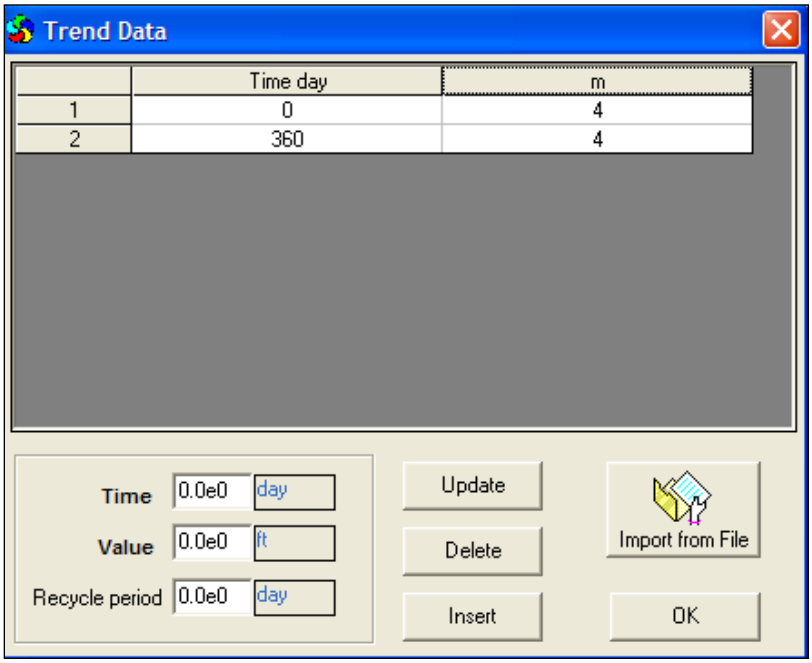


The 'Trend' window appears. Here you can adjust the nominal value for the head, the periodic function that describes head transients, and any randomness associated with the head values. You can explore these values seen here, but at a minimum you need to set the nominal head equal to value that you had assigned for the Constant Head. For this example, use the value of 3 feet.



Step 7: Click the 'Edit' button next to the 'Data points' field.

The 'Trend Data' window appears. Notice the default head is set to 4 m (in the right column).



Step 8: In the first row, click on the number '4' that corresponds to time 0.

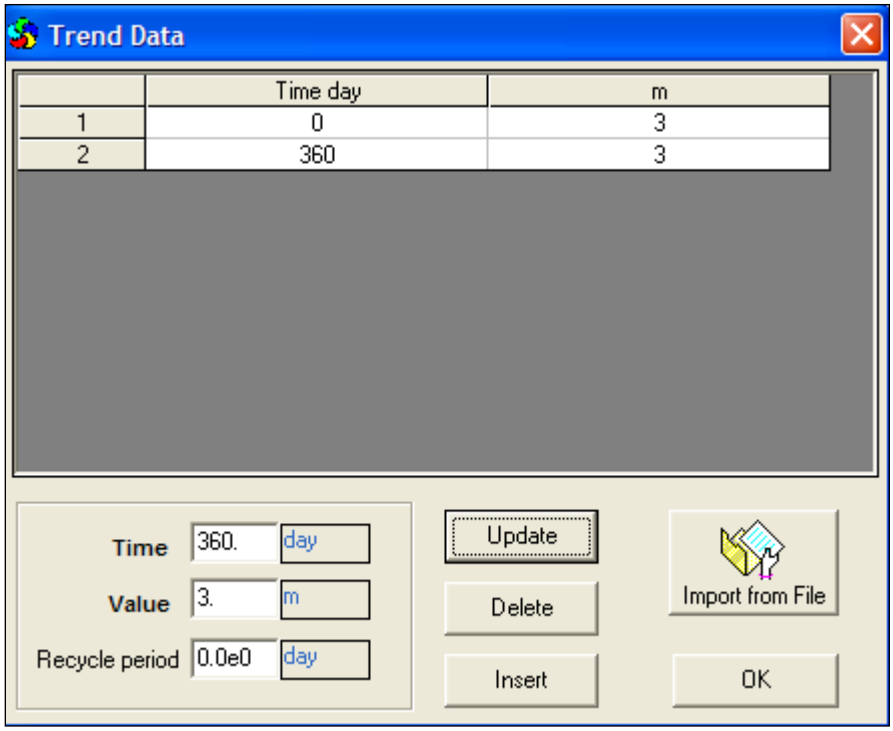
Step 9: In the ‘Value’ field, change the units to meters (m) and enter the value of ‘3’.

Step 10: Click the ‘Update’ button.

Step 11: In the second row , click on the ‘4’ that corresponds to time 360.

Step 12: In the ‘Value’ field, change the units to meters (m) and enter the value of ‘3’.

Step 13: Click the ‘Update’ button.

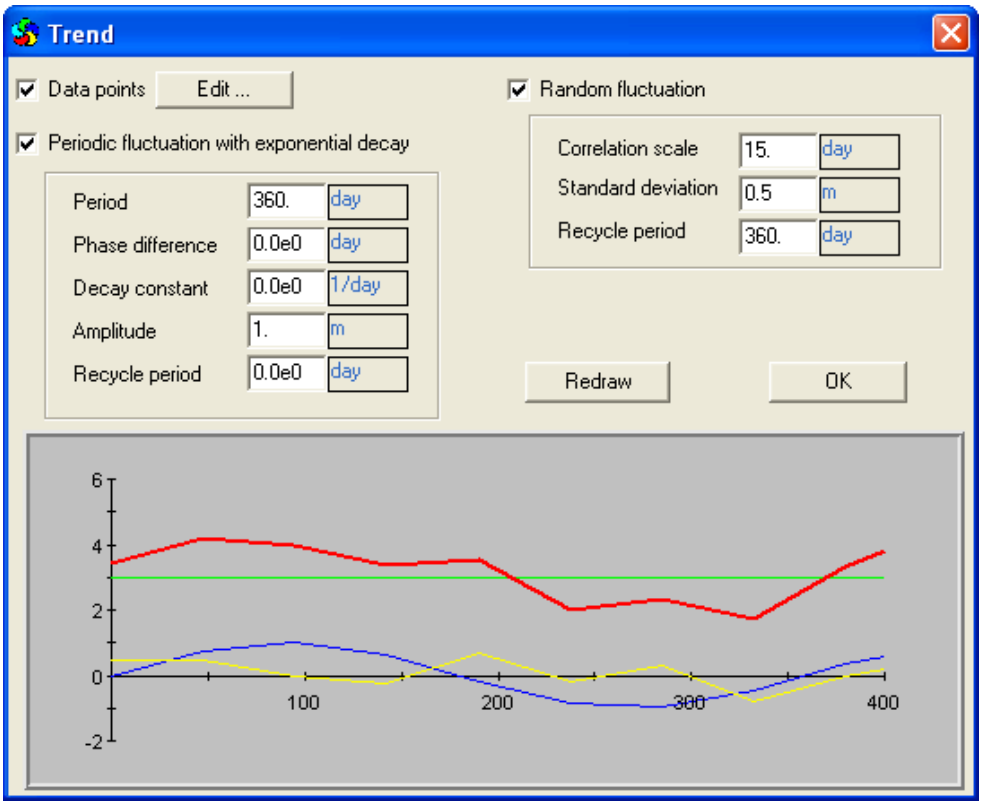


Step 14: Click the ‘OK’ button.

You return to the ‘Trend’ window.

Step 15: Click the ‘Redraw’ button.

The function displayed in the lower half of the ‘Trend’ window now updates to reflect the nominal value (3 meters) as the green line, the periodic transients as the blue line, the random fluctuations as the yellow line, and the overall transient response as the red line.



Step 16: Click the 'OK' button.

Step 2 through Step 16 should be repeated for any other layers that the head-dependent body of water affects.

Next, you should solve the model in steady state to provide some initial (baseline) conditions for the transient simulation.

Step 17: Discretize the model.

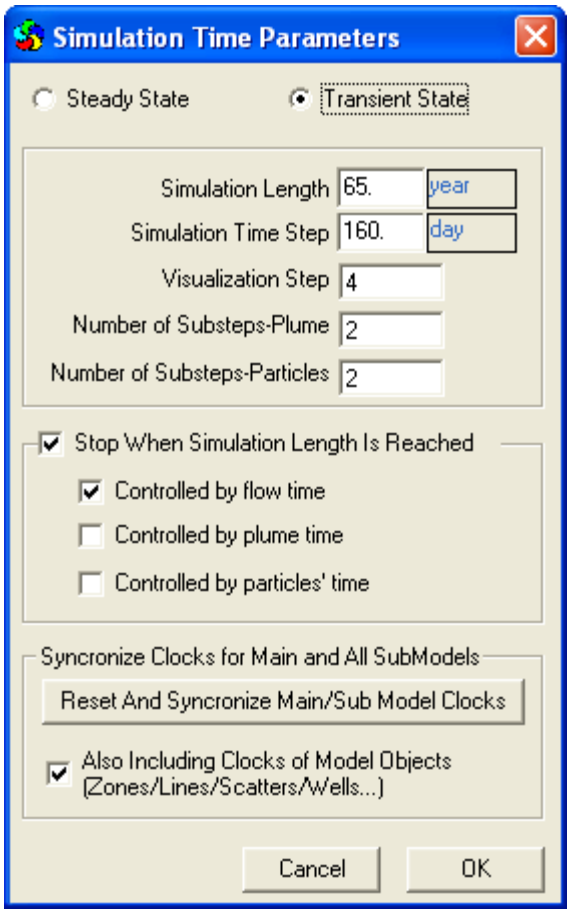
Step 18: Click the 'Forward' button to solve the model.

Step 19: Click the 'Pause' button to stop the simulation.

The presence of particles makes the model continue translating the particles, but you will need to stop it before switching to transient conditions, for the sake of a more clear view of the results.

Step 20: Click the 'Set Simulation Time Parameters' button, located at Row 6, Column 1 of the button palette.

The 'Simulation Time Parameters' window appears.

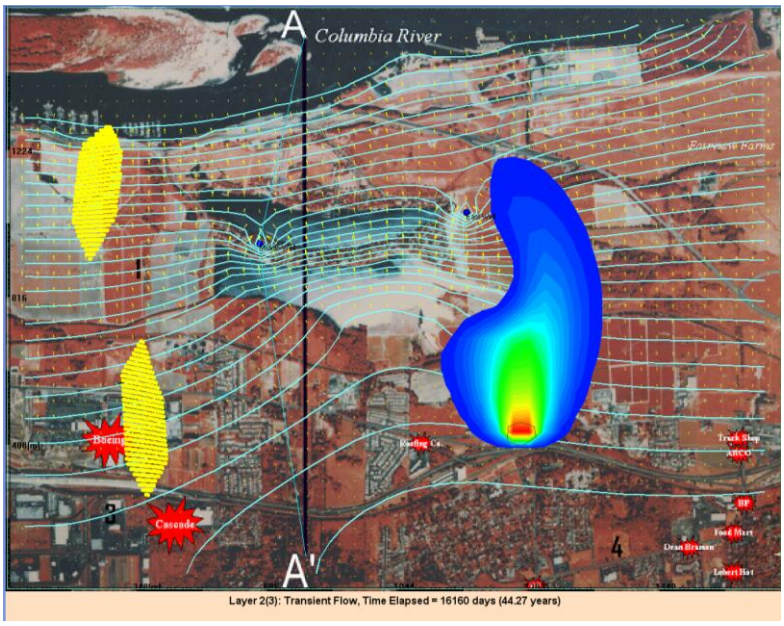


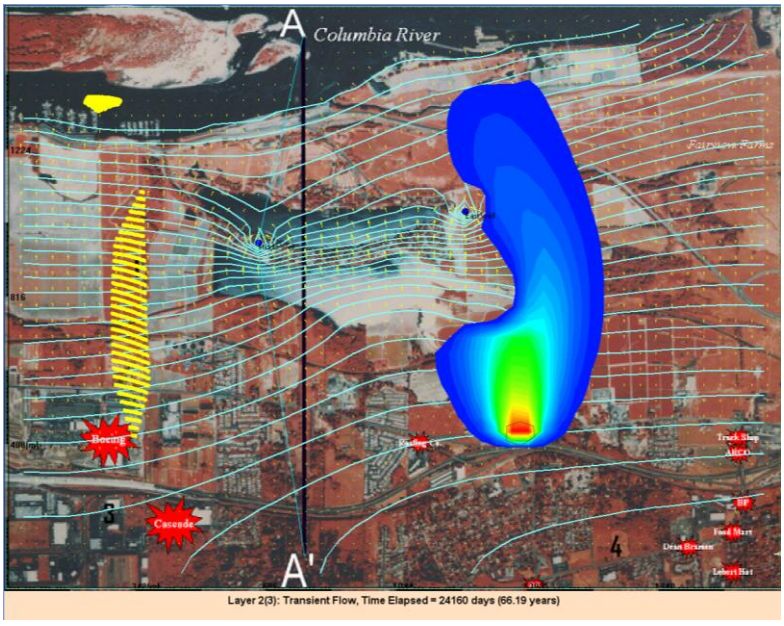
Step 21: Select 'Transient State'.

Step 22: Click the 'OK' button.

Step 23: Click the 'Forward' button to solve the model.

You can observe the working area of IGW. With every visualization step the head and velocity fields are updated. The particle motion (contaminant transport) is also affected by changing head and velocity fields. A snapshot of transient simulation after 44 years and 66 years is shown below. Notice that the plume reaches close to East Well. Also notice that the shape of the plume is different from the one before.





Step 24: Click the 'Pause' button to stop the simulation.

CHAPTER 19: CONCLUSION

This tutorial was designed to move you from the realm of IGW beginner to that of IGW user. After completing all of the sections you should have a good idea of the capabilities of the software. However, this tutorial is in no way exhaustive. There are many more features that are not even mentioned in this document. Work continues on the IGW software and all of the associated material. You should consult the User's Manual and program Help file for more information, and stay alert for periodic software and documentation updates.

Thank you for taking the time to examine IGW Version 5.0. We hope you find this software to be a powerful and empowering tool in your hydrogeologic endeavors.

Button Palette:

Create New Project (1, 1)

Clicking this button opens a new model without saving the current work. It performs the same function as the 'New Model' operation in the **File** menu

Example User Manual: Section 3.3.1

Open Model (1, 2)

Clicking this button allows the user to open a previously-saved model in IGW.

Example User Manual:

Save Model (1, 3)

Clicking this button allows the user to save the current model into any desired location

Example User Manual:

Define model domain/Import Basemap (1, 4)

Clicking this button initiates the process of setting a picture file as a basemap in the Working Area by opening the 'Model Scale and Basemap' window.

Example: Domain, Basemap User Manual: Chapter 5

Reset Toolbar (2, 1)

Clicking this button resets the cursor from any previous state such as 'help mode' or 'draw mode'. The cursor is initialized to select new buttons or perform other functions

Example User Manual:

Create Zone/Assign Properties (2, 2)

Clicking this button allows the user to define a zone (polygon) within the Working Area. The cursor is set to 'draw mode' and the user may simply click within the Working Area to define points that denote the outline of the zone.

Example User Manual: Chapter 7

Create a new polyline and assign property (2, 3)

Clicking this button allows the user to define a polyline (a series of line segments) within the Working Area. The cursor is set to 'draw mode' and the user may simply click within the Working Area to define points that correspond to the line-segment endpoints.

Example User Manual: Chapter 8

Add Well (2, 4)

Clicking this button allows the user to define a well (point) within the Working Area. The cursor is set to 'draw mode' and the user may simply click within the Working Area to define a point that corresponds to the location of the well.

Example User Manual: Chapter 9

Modify Existing Zone (3, 1)

Clicking this button allows the user to replace the active zone with another zone, without having to redefine the zone attributes or any associated scatter points. The first click of the button brings up one of two windows: 1) a 'Message' window appears with the text, 'You should select a zone first!' if no zone is currently selected; or 2) a 'Warning' windows appears with the text, 'Are you sure that you want to replace the current zone with a new one?', if there is an active zone.

If the 'Message' window appears, the user should click 'OK', select a zone and then re-select the 'Redefine Applied Area for a Zone' button.

If the 'Warning' window appears, verify that the correct zone is selected. If not, then select 'No', activate the desired zone, click the 'Reset toolbar buttons state' button, then re-select the 'Redefine Applied Area for a Zone' button. If the correct zone is selected, then click 'Yes'. The cursor enters 'draw mode' and the user may define a completely new zone to replace the old one. Once draw mode has been entered, the user may continually replace the previous zone until satisfied.

Example User Manual: Section 7.5

Select/Edit Zone (3, 2)

Clicking this button allows the user the select a zone within the Working Area. The cursor is set to 'select mode' and the user may simply click within a zone in the Working Area to select it. This is alternatively referred to as 'making the zone active'. When a feature is selected it appears outlined in red in the Working Area and highlighted in AE window.

Example User Manual: Section 7.2

Select/Edit Polyline (3, 3)

Clicking this button allows the user the select a polyline within the Working Area. The cursor is set to 'select mode' and the user may simply click on a polyline in the Working Area to select it. This is alternatively referred to as 'making the polyline active'. When a feature is selected, it appears outlined in red in the Working Area and highlighted in the AE window.

Example User Manual: Section 8.2

Select/Edit Well (3, 4)

Clicking this button allows the user the select a well within the Working Area. The cursor is set to 'select mode' and the user may simply click on a well in the Working Area to select it. This is alternatively referred to as 'making the well active'. When a feature is selected it appears outlined in red in the Working Area and highlighted in the AE window.

Example User Manual: Section 9.2

Add Single Particle (4, 1)

Clicking this button allows the user to add a single particle in the Working Area. The cursor is set to 'draw mode' and the user may simply click within the Working Area to define a point that corresponds to the desired location of the particle.

Example User Manual: Chapter 10

Add Particles Inside Polygon (4, 2)

Clicking this button allows the user to add a group of particles in the Working Area. The cursor is set to 'draw mode' and the user may simply define a zone that outlines the desired location of the particles within the Working Area. Once the zone is defined, the 'Particles' window appears prompting the user

to enter the number of particle columns desired. Once the number is entered, either click 'OK' to create the particle zone or 'Cancel' to abort.

Example User Manual: Chapter 10

Add Particles Along Polyline (4, 3)

Clicking this button allows the user to add a polyline (a series of line segments) of particles in the Working Area. The cursor is set to 'draw mode' and the user may simply define a polyline that indicates the desired location of the particles within the Working Area. Once the polyline is defined, the 'Particles' window appears prompting the user to enter the number of particles to be released along the polyline. Once the number is entered click 'OK' to create the particle polyline or 'Cancel' to abort.

Example User Manual: Chapter 10

Add Particles Around Well(s) (4, 4)

Clicking this button allows the user to add particles to existing wells.

Example User Manual: Section 10.1.4; Chapter 10

Add Scatter Point (5, 1)

Clicking this button allows the user to add a scatter point (associated with a zone) to the Working Area. The cursor is set to 'draw mode' and the user may click at any point in the Working Area to define a scatter point. This process may be repeated as desired without having to re-select the 'Add Scatter Point' button (however, be sure to wait for the crosshair cursor to reappear before defining another scatter point).

If no zone is active, attempting to add scatter points in the Working Area will bring up the 'Message' window with the text 'You should select a zone first!'. In this case, click the 'OK' button, select the desired zone, and then re-click the 'Add Scatter Point' button.

Example User Manual:

I Before adding scatter points, the user should first select the polygon onto which they will add the points. In order to access to scatter points, the corresponding zone should be activated.

Select Scatter Point (5, 2)

Clicking this button allows the user to select a scatter point within the Working Area. The zone associated with the desired scatter point should be active prior to clicking this button, or else the user will not be able to select the desired scatter point. The cursor is set to 'select mode', and the user may simply click on a scatter point (associated with the active zone) in the Working Area to select it. This is alternatively referred to as 'making the scatter point active'. When a feature is selected, it appears outlined in red in the Working Area and highlighted in the AE window.

If no zone is active (or none are yet defined in the model – in which case no scatter points can exist yet and therefore trying to select one does not make much sense) then attempting to select a scatter point elicits the 'Warning' window with the text 'You should select a zone first!'. In this case click 'OK', select the zone associated with the desired scatter point, and re-click the 'Select Scatter Point' button.

Example User Manual: Section 7.7

Add 3D Attribute Model (5, 3)

This feature allows the user to treat the entire model by adding recharge to the first active layer within

each node of the simulation for which these boundaries are established.

For instance, it could be that the water table is not present until layer 3 in the model, and therefore it makes sense to only apply recharge to this layer versus the top layer for best results. The user outlines a portion of the active area (or the entire active area if so desired) to apply this feature, and once discretized the model will add active recharge/evaporation only to the active layers

Example User Manual:

Select 3D Attribute Model (5, 4)

Clicking this button allows the user to select any of the 3D attribute sub-models that were created in the above step for adding recharge to an active layer of the simulation. This feature is identical to selecting a zone within a model.

Example User Manual:

Set Simulation Time Parameters (6, 1)

Clicking this button allows the user to edit the time parameters associated with the model by opening the 'Simulation Time Parameters' window.

Example User Manual: Chapter 11

Set/Edit Default Parameters for the Active Model (6, 2)

Clicking this button allows the user to change the model parameters of the active modeling layer assigned to zones when they are created by opening the 'Default Attribute' window.

Example User Manual: Chapter 6

Define Cross-section (6, 3)

Clicking this button allows the user to create a cross-section by defining its extent (as a series of line segments) within the Working Area. The cursor is set to 'draw mode' and the user may simply click within the Working Area to define points that correspond to the desired cross-section line-segment endpoints.

Example User Manual: Chapter 16

Select Cross-section (6, 4)

Clicking this button allows the user to select a cross-section within the Working Area. The cursor is set to 'select mode' and the user may simply click on a cross-section in the Working Area to select it. This is alternatively referred to as 'making the cross-section active'. When a feature is selected it appears outlined in red in the Working Area and highlighted in the AE window.

Example User Manual: Section 16.2

Deep Discretization (7, 1)

Clicking this button allows the user to adjust the nodal grid by opening the 'Define Model Grid' window.

Example User Manual: Chapter 12

Shallow Discretization (7, 2)

Clicking this button applies the changes made in a conceptual model onto the numerical model (also referred to as 'discretizing the changes').

Example User Manual: Chapter 12

Create Submodel (7, 3)

Clicking this button allows the user to define a submodel (polygon) within the Working Area. The cursor is set to 'draw mode', and the user may simply click within the Working Area to define points that denote the outline of the desired submodel.

Example User Manual: Chapter 15

Select/Edit Submodel (7, 4)

Clicking this button allows the user to select a submodel within the Working Area. The cursor is set to 'select mode' and the user may simply click within a submodel in the Working Area to select it. This is alternatively referred to as 'making the submodel active'. When a feature is selected it appears outlined in red in the Working Area and highlighted in the AE window.

Example User Manual: Section 15.3

Display Options (8, 1)

Clicking this button allows the user to adjust numerous display parameters by opening the 'Display Options for Model 1' window. This is the same window that appears when 'Option…' from the 'Display' menu is selected.

Example User Manual: Section 19.1

Refresh Screen (8, 2)

Clicking this button causes all IGW Version 5.0P screens and windows to be redrawn with any incorporated changes such as window resizing or a changing of color for a certain attribute. It is not always necessary to click the 'Refresh Screen' button, as the software automatically updates most changes. Clicking this button is the same as selecting 'Refresh' on the 'Display' menu.

Example User Manual: Section 3.3.6, 19.3

Zoom In (8, 3)

Clicking this button enlarges the Working Area and Working Area Attribute Display within the Model Screen. This is the same as selecting 'Zoom in' on the 'Display' menu.

Example User Manual: Section 3.3.6, 19.4

Zoom Out (8, 4)

Clicking this button shrinks the Working Area and Working Area Attribute Display within the Model Screen (see Section 3.7). This is the same as selecting 'Zoom out' on the Display menu.

Example User Manual: Section 3.3.6, 19.4

Numerical Solver Settings (9, 1)

Clicking this button allows the user to adjust flow, transport, and stochastic model solver settings by opening the 'Solver' window.

Example User Manual: Chapter 13

Run Model Backward (9, 2)

Clicking this button causes the software to track particles in the opposite direction of the velocity vectors. If the button is inactive it will appear grayed out:

Example User Manual: Section 10.3.2

Pause/Stop Model (9, 3)

Clicking this button causes the software to stop the present simulation at the current state. The software will finish its calculations and the model redraws for the present time step before appearing idle

Example User Manual:

Run Model Forward (9, 4)

Clicking this button causes the software to solve the numerical model. If the model is set to transient state, or there are transport calculations to be done in a steady-state model, then the software will continually update as it proceeds through the simulation. Options for running the model are discussed in Chapter 14. The solver options are presented in Chapter 13. The model must be discretized (see Chapter 12) before attempting to solve, or else an error message will appear.

Example User Manual: Chapter 13, 14 and 12

Reset Flow Clock (10, 1)

Clicking this button resets the 'Flow Time' display in the Step Adjustment and Time Display Interface, and the flow component of the elapsed time displayed in the Working Area Attribute Display.

Example User Manual:

Reset Concentration Clock (10, 2)

Clicking this button resets the 'Plume Time' display in the Step Adjustment and Time Display Interface, and the plume component of the elapsed time displayed in the Working Area Attribute Display

Example User Manual:

Initialize Plume (10, 3)

Clicking this button returns all concentration plumes to their original locations and resets the concentration values for all cells in the model.

Example User Manual: Section 14.3

Reset Particle Clock (10, 4)

Clicking this button resets the 'Particle Time' display in the Step Adjustment and Time Display Interface, and the particle component of the elapsed time displayed in the Working Area Attribute Display.

Example User Manual: Section 3.4

Add Text (11, 1)

Clicking this button allows the user to add a text field in the Working Area. The cursor is set to 'draw mode', and the user may simply click at a point in the Working Area to designate a text field.

Example User Manual:

> **I** The clicked point corresponds to the upper-left-hand corner of the text field. Also, the text field will not be visible until text is actually typed into it via the AE window (**Section 4.1**).

Select Text (11, 2)

Clicking this button allows the user the select a text field within the Working Area. The cursor is set to 'select mode' and the user may simply click on a text field in the Working Area to select it. This is alternatively referred to as 'making the text field active'. When a feature is selected, it appears outlined in red in the Working Area and highlighted in the AE window.

Example User Manual: Section 19.2

Initialize Particle(s) (11, 3)

Clicking this button returns all particles to their original locations.

Example User Manual: Chapter 10

Delete Particle(s) (11, 4)

Clicking this button deletes all particles from the conceptual model.

Example User Manual: Chapter 10

No Capture (12, 1)

Clicking this button turns off all capture options.

Example User Manual: Section 23.1.2

External Calling Capture (12, 3)

Clicking this button activates the time-step save feature and allows the user to invoke the manual timer function.

Example User Manual: Section 23.1.2

Timing Capture (12, 3)

Clicking this button activates the automatic timing screen capture option.

Example User Manual: Section 23.1.2

Set Capture Option (12, 4)

Clicking this button allows the user to edit screen capture options by opening the 'Automatic Capture' window.

Example User Manual: Section 23.1.2

3.7 Model Screen / Working Area / Attribute Display

The remainder of the Main Window is referred to as the 'Model Screen'. It is pictured in **Figure 3.7.1.**

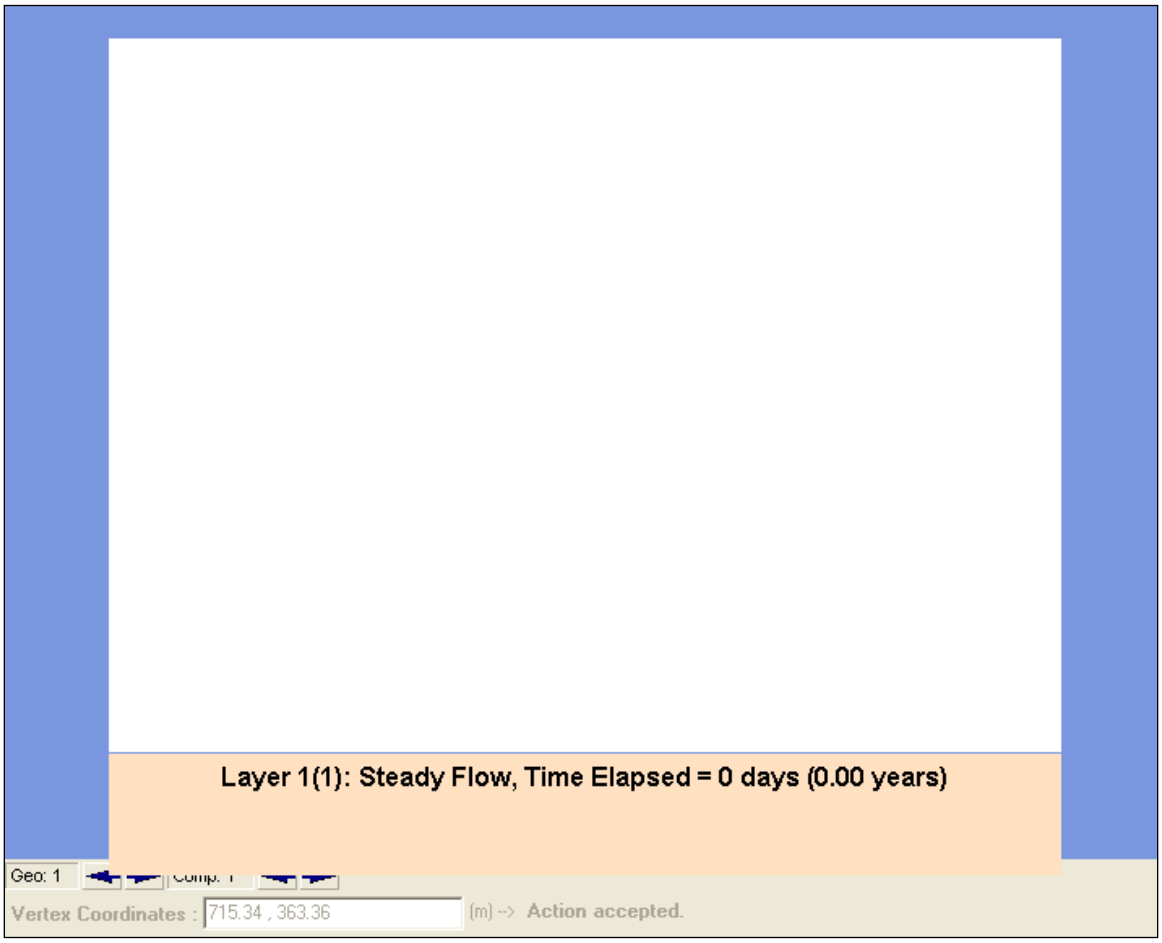


Figure 3.7.1 The Model Screen

The Model Screen exists mainly to provide a background against which the 'Working Area' (the large white rectangle) and the 'Working Area Attribute Display' (WAAD, the peach colored rectangle) can be displayed.

A secondary feature of the Model Screen is the 'Vertex Coordinate Interface' (VCI). It is located directly below the dark gray portion of the Model Screen. It displays the coordinates of the last vertex defined, and allows the user to manually enter the coordinates of features when defining them in the Working Area. This provides for much greater accuracy compared to pointing and clicking with the mouse. The user should enter the coordinates in simple Cartesian format by adding a ',end' for the final vertex. The VCI text will appear black (it is gray otherwise) when the cursor is in 'draw mode'

I The 'end' command is not necessary when defining point features such as wells or single particles. Also, using the ',end' command to describe the feature before enough vertices [or points] have been defined, will result in the feature defining process being aborted. Certain feature-related status messages will also appear to the right of the VCI field (such as 'Action accepted' or 'Action accepted and ended'). Instead of using the 'end' command, the user may also double-click to finalize drawing a polyline.

The two slender gray boxes at the bottom of the Model Screen are known as the 'Left Message Area' (LMA), and the 'Right Message Area' (RMA), respectively. These will display messages concerning the model solution status.

The Working Area is the region where the conceptual modeling is performed and subsequent solutions are obtained. It can be displayed anywhere within the Model Screen, and resized using the 'Zoom in' and 'Zoom out' buttons. It is not restricted by the size of the model screen: if it is larger than the model screen edges will appear to go behind the other software interface components (i.e. the CAT, Button Palette, SATDI, VCI, LMA / RMA, etc.), and even off the edges of the monitor.

The WAAD is attached to the bottom of the Working Area. It displays information concerning the type of flow and elapsed time of the current simulation. The WAAD will remain proportional in size to the Working Area. The user may click on the title in the WAAD to open the 'Input title' window, and subsequently type in extra text to be added in front of the default title. The user may click on any of the three empty lines below the title to open the 'Input note' window, and subsequently type in text to be displayed below the title line.

3.8 The Cursor

The cursor is important to using IGW Version 4.7 because of the software's graphical interface. The cursor has a number of modes (discussed in **Section 3.3)** that it may enter, depending on the current status of the software. These include:

- **Default mode**: The default cursor mode. The cursor always appears as its default shape. This mode is used for performing basic Windows™ functions and using the buttons and menus in the software. In order to initialize the cursor, the default mode should be used.

- **Draw and add text mode**: This mode is used when defining features and adding text in the Working Area. The cursor appears as a crosshair when it is positioned in the Working Area.

- **Select mode**: This mode is used when selecting features in the Working Area. The cursor appears with a question mark next to it when positioned in the Working Area.

- **Node edit mode**: This mode is used when 'Node Edit' is selected from the right-click menu (see **Section 3.9**). The cursor appears with a question mark next to it (when in the Working Area), and changes to a large crosshair when positioned at the vertices or line segment midpoints of a selected feature. (This mode may still be active when no feature is selected, but the cursor does not change in this case). When in this mode, selected features may be redefined by 1.) Clicking and dragging vertices (appearing as black squares), or 2.) Defining new vertices by clicking and dragging on line segment midpoints (appearing as blue crosses). If a vertex is moved so that it lies directly between two other vertices, the software will automatically eliminate that vertex (as it is redundant and no longer necessary). See the sections on redefining features (**Section 7.3** for zones and **Section 8.3** for polylines) for more information.

> **I** When the model is running, the cursor turns into an hourglass.